QuickField™

Finite Element Analysis System

Version 4.2T

User's Guide

Tera Analysis

Tera Analysis Ltd.
Knasterhovvej 21
DK-5700 Svendborg
Denmark

Telephone:	+45 6354 0080	
Fax:	+45 6254 2331	
E-mail:	sales@quickfield.com	(sales and distribution)
	support@quickfield.com	(technical support)
WEB:	http://www.quickfield.com	

Contents

About This Manual

What Is QuickField?

Welcome to QuickField Finite Elements Analysis System. QuickField is a PC-oriented interactive environment for electromagnetic, thermal and stress analysis. Standard analysis types include:

- Electrostatics.
- Linear and nonlinear magnetostatics.
- Time-harmonic magnetics (involving eddy current analysis).
- Linear and nonlinear, steady state and transient heat transfer and diffusion.
- Linear stress analysis.
- Coupled problems.

During a 15-minute session, you can describe the problem (geometry, material properties, sources and other conditions), obtain solution with high accuracy and analyze field details looking through full color picture. With QuickField, complicated field problems can be solved on your PC instead of large mainframes or workstations.

How to Use this Manual

This manual has nine chapters:

Chapter 1, *"Getting Started",* describes first steps of using QuickField. In this chapter, you will learn how to install and start the package.

Chapter 2, "*Introductory Guide",* briefly describes the organization of QuickField and gives an overview of analysis capabilities.

Chapter 3, *"Problem Description"*, explains how to specify the analysis type and general problem features.

Chapter 4, *"Model Geometry Definition"*, explains how to describe geometry of the model, build the mesh, and define material properties and boundary conditions.

Chapter 5, *"Problem Parameters Description"*, introduces non-geometric data file organization, and the way to attach this file to the model.

Chapter 6, *"Solving the Problem"*, tells you how to start the solver to obtain analysis results.

Chapter 7, *"Analyzing Solution"*, introduces QuickField Postprocessor, its features and capabilities.

Chapter 8, *"Theoretical Description"*, contains mathematical formulations for all problem types that can be solved with QuickField. Read this chapter to learn if QuickField can solve your particular problem.

Chapter 9, *"Examples"*, contains description of some example problems, which can be analyzed using QuickField.

Conventions

In this manual we use SMALL CAPITAL LETTERS to specify the names of keys on your keyboard. For example, ENTER, ESC, or ALT. Four arrows on the keyboard, collectively named the DIRECTION keys, are named for the direction the key points: UP ARROW, DOWN ARROW, RIGHT ARROW, and LEFT ARROW.

A plus sign (+) between key names means to hold down the first key while you press the second key. A comma (,) between key names means to press the keys one after the other.

Bold type is used for QuickField menu and dialog options.

CHAPTER 1

Getting Started

Required Hardware Configuration

Computer:	Personal computer with a 486 or higher processor.
Operating System:	Microsoft Windows 95 or Microsoft Windows NT 4.0 Service Pack 3 or later
Memory:	8MB minimum, 32MB recommended. Additional memory can improve performance for very huge problems.
Video:	VGA or higher-resolution video adapter (Super VGA, 65536-color recommended).
Mouse:	Microsoft Mouse or compatible pointing device.
Peripherals:	A parallel port.

QuickField Installation

To install QuickField software on your computer:

1. Insert disk labeled “QuickField Disk 1” into your floppy disk drive.
2. Bring up Control Panel and double-click **Add/Remove Programs**.
3. Follow the instructions on the screen.

Now you can start QuickField from your Start menu.

When you start QuickField for the first time you will be requested to enter a password. The password is a 16-character length sequence of letters given to you on a sheet of paper with your QuickField package. The password is stored in the Password.txt file in your QuickField program files folder and will be used in all subsequent sessions. To change the password you can edit this file or make use of the **Password** command in the **Edit** menu, while all document windows in QuickField are closed.

Note. To solve very large problems on a computer with insufficient memory it is essential that virtual memory is configured optimally.

To manage virtual memory settings:

1. Bring up Control Panel and double-click **System**.
2. Switch to **Performance** tab.
3. See Windows Help for details.

Uninstalling QuickField

To remove QuickField software from your system:

1. Bring up Control Panel and double-click **Add/Remove Programs**.
2. Select QuickField in the list of installed software.
3. Click **Add/Remove**.

CHAPTER 2

Introductory Guide

This chapter briefly describes the basic organization of the QuickField program. It presents an overview of the available capabilities.

The aim of this chapter is to get you started with modeling in QuickField. If you are new to the QuickField, we strongly recommend you to study this chapter. If you haven't yet installed QuickField, please do so. For information on installing QuickField see Chapter 1.

Basic Organization of QuickField

In QuickField, you work with several types of documents: problems, geometry models, material libraries and so on. Each document is opened into a separate window within the main application window of QuickField. You can open any number of documents at once. When switching between windows, you switch from one document to another. Only one document and one window are active at a time, so you can edit the active document. Editing actions are listed in the menu residing on the top of main window of QuickField. Menu contents are different for different document types. You can also use context-specific menus, which are available by right-button mouse click on specific items in document window.

The QuickField documents are:

Problem corresponds to specific physical problem solved by QuickField. This document stores the general problem parameters, such as the type of analysis (“Electrostatics”, “Magnetostatics”, “Heat transfer” and etc.) or the model type (planar or axisymmetric). The detailed description of working with problems is given in Chapter 3.

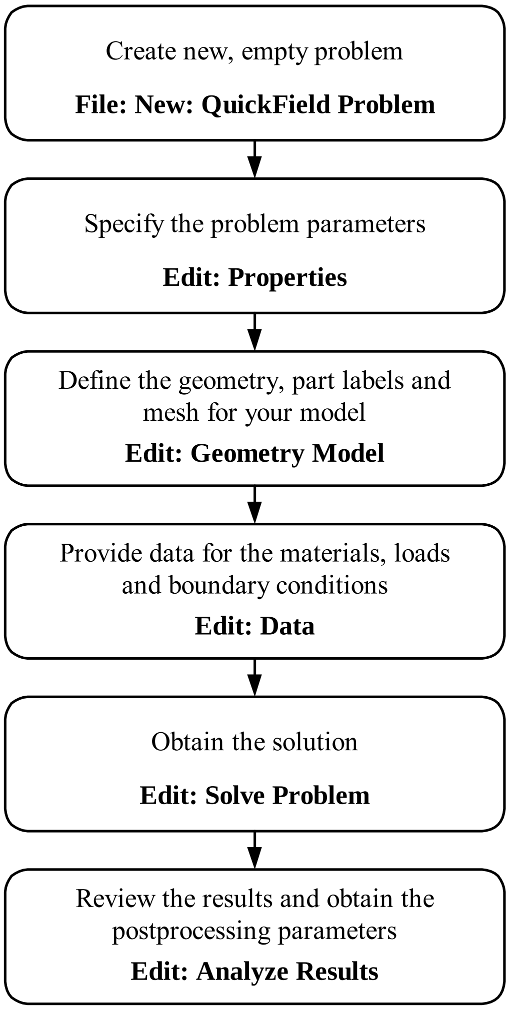
Geometric Model is a complete description of the geometry, the part labels and the mesh of your model. Several problems may share the same model (this is particularly useful for coupling analysis). Editing models is described in details in Chapter 4.

Property Description, or Data documents are specific to types of analysis (Electrostatics data, Stress Analysis data, etc.) These documents store the values of material properties, loadings and boundary conditions for different part labels. Data documents can be used as material libraries for many different problems. The detailed description of how to specify material properties and boundary conditions is given in Chapter 5.

For the problem to be solved and analyzed, it must reference the model and data documents. For convenience, the problem can reference two data documents at once: one document containing properties for commonly used materials (material library), and another document containing data specific for the problem or group of problems.

Between sessions, QuickField documents are stored in disk files, separate file for each document. During the session, you can create new documents or open existing ones. The detailed description of how to get and explore the results of the analysis is given in Chapter 6 and Chapter 7.

Using this very flexible architecture, QuickField helps you build and analyze your design problems very quickly. In analyzing a problem, the typical sequence of phases that you go through with QuickField is depicted in the flowchart below:



Window Management Tips

QuickField is a Multiple Documents Interface (MDI) application, so you can work with several QuickField documents (problem descriptions, geometry models, data sets, etc.) at once. Dealing with specific documents will be discussed later in dedicated chapters, and now let's discuss general principles of creating new documents, opening existing ones, switching between them and so on.

To create a new, empty QuickField document, you can:

- Launch QuickField from your **Start** menu, and then click **New** in the QuickField's **File** menu or use correspondent icon on QuickField toolbar. The dialog box appears asking you, which kind of QuickField document you want to create; or
- Click right mouse button on your desktop (screen region not occupied by any application), and choose the kind of document in the context menu (thus creating the document on the desktop); or
- Use Windows Explorer to select specific directory where you want new document to be created, and click **New** in the main menu or context menu of Explorer.

To open an existing QuickField document, you can:

- Double-click it in Windows Explorer or any file management utility; or
- While QuickField is running, click **Open** in the QuickField's **File** menu or click **Open** tool on QuickField's toolbar; or
- Drag the document's icon from Explorer to any part of the QuickField window.

To close the document, click **Close** in the **File** menu, or click **Close** icon on the document's window frame. If the document has been changed since last save, you will be asked to save the changes to file.

To switch between windows within QuickField, press CTRL+TAB or click on any visible part of the window you want to switch to.

Once the document is open, its window can be minimized to an icon, maximized to the full size of QuickField window, or brought to 'normal' size, which you can change by dragging the window's corners. This is particularly useful when you want to see several documents at once. Also you can automatically arrange all non-minimized windows side-by-side by clicking **Tile Horizontally** or **Tile Vertically** in the **Window** menu.

Some windows can be split up to four separate panes. To split window you can point the splitter box – small gray box appears at the top of the vertical scroll bar or at the left side of the horizontal one. When the mouse pointer changes its form drag the splitter bar to the position you want. You can use also **Split Window** command in the **Windows** menu. To switch between panes simply click the desired one or press F6 key.

To return to a single window double click the split bar or point it and drag until it disappears.

Overview of Analysis Capabilities

This section provides you with the basic information on different analysis capabilities. For detailed formulations of these capabilities see Chapter 8.

Magnetostatic Analysis

Magnetic analysis is used to design or analyze variety of devices such as solenoids, electric motors, magnetic shields, permanent magnets, magnetic disk drives, and so forth. Generally the quantities of interest in magnetostatic analysis are magnetic flux density, field intensity, forces, torques, inductance, and flux linkage.

QuickField can perform linear and nonlinear magnetostatic analysis for 2-D and axisymmetric models. The program is based on a vector potential formulation. Following options are available for magnetic analysis:

Material properties: air, orthotropic materials with constant permeability, ferromagnets, current carrying conductors, and permanent magnets. B-H curves for ferromagnets can easily be defined through an interactive curve editor, see the *“Editing the Curves”* section in Chapter 5.

Loading sources: current or current density, uniform external field and permanent magnets.

Boundary conditions: Prescribed potential values (Dirichlet condition), prescribed values for tangential flux density (Neumann condition), constant potential constraint for zero normal flux conditions on the surface of superconductor.

Postprocessing results: magnetic potential, flux density, field intensity, forces, torques, magnetic energy, flux linkage, self and mutual inductances.

Special features: A postprocessing calculator is available for evaluating user-defined integrals on given curves and surfaces. The magnetic forces can be used for stress analysis on any existing part (magneto-structural coupling) . A self-descriptive inductance wizard is available for easily calculation of self- and mutual inductance of your coils.

Time-Harmonic Electromagnetic Analysis

Time-harmonic electromagnetic analysis is used to analyze magnetic field caused by alternating currents and, vise versa, electric currents induced by alternating magnetic field (eddy currents). This kind of analysis is useful with different inductor devices, solenoids, electric motors, and so forth. Generally the quantities of interest in harmonic magnetic analysis are electric current (and its source and induced component), voltage, generated Joule heat, magnetic flux density, field intensity, forces, torques, impedance and inductance.

Following options are available for harmonic magnetic analysis:

Material properties: air, orthotropic materials with constant permeability, current carrying conductors with known current or voltage.

Loading sources: voltage, total current, current density, uniform external field.

Boundary conditions: Prescribed potential values (Dirichlet condition), prescribed values for tangential flux density (Neumann condition), constant potential constraint for zero normal flux conditions on the surface of superconductor.

Postprocessing results: magnetic potential, current density, voltage, flux density, field intensity, forces, torques, Joule heat, magnetic energy, impedances, self and mutual inductances.

Special features: A postprocessing calculator is available for evaluating user-defined integrals on given curves and surfaces. The magnetic forces can be used for stress analysis on any existing part (magneto-structural coupling); and power losses can be used as heat sources for thermal analysis (electro-thermal coupling). Two self-descriptive wizards are available: one of them for easily calculation of mutual and self inductance of your coils and the second for calculation of the impedance.

Electrostatic Analysis

Electrostatic analysis is used to design or analyze variety of capacitive systems such as fuses, transmission lines and so forth. Generally the quantities of interest in electrostatic analysis are voltages, electric fields, capacitances, and electric forces.

QuickField can perform linear electrostatic analysis for 2-D and axisymmetric models. The program is based on Poisson's equation. Following options are available for electrostatic analysis:

Material properties: air, orthotropic materials with constant permittivity.

Loading sources: Voltages, and electric charge density.

Boundary conditions: Prescribed potential values (Voltages), prescribed values for normal derivatives (surface charges), and prescribed constraints for constant potential boundaries with given total charges.

Postprocessing results: voltages, electric fields, gradients of electric field, flux densities (electric displacements), surface charges, self and mutual capacitances, forces, torques, and electric energy.

Special features: A postprocessing calculator is available for evaluating user-defined integrals on given curves and surfaces. Floating conductors with unknown voltages and given charges can be modeled. The electrostatic forces can be used for stresses on any existing part (electro-structural coupling).. A self-descriptive capacitance wizard is available for easily calculation of self- and mutual capacitance of your conductors.

Current Flow Analysis

Current flow analysis is used to analyze variety of conductive systems. Generally the quantities of interest in current flow analysis are voltages, current densities, electric power losses (Joule heat).

QuickField can perform linear current flow analysis for 2-D and axisymmetric models. The program is based on Poisson's equation. Following options are available for current flow analysis:

Material properties: orthotropic materials with constant resistivity.

Loading sources: Voltages, electric current density.

Boundary conditions: Prescribed potential values (Voltages), prescribed values for normal derivatives (surface current densities), and prescribed constraints for constant potential boundaries.

Postprocessing results: voltages, current densities, electric fields, electric current through a surface, and power losses.

Special features: A postprocessing calculator is available for evaluating user-defined integrals on given curves and surfaces. The electric power losses can be used as heat sources for thermal analysis (electro-thermal coupling).

Thermal Analysis

Thermal analysis plays an important role in design of many different mechanical and electrical systems. Generally the quantities of interest in thermal analysis are temperature distribution, thermal gradients, and heat losses. Transient analysis allows you to simulate transition of heat distribution between two heating states of a system.

QuickField can perform linear and nonlinear thermal analysis for 2-D and axisymmetric models. The program is based on heat conduction equation with convection and radiation boundary conditions. Following options are available for thermal analysis:

Material properties: orthotropic materials with constant thermal conductivity, isotropic temperature dependent conductivities, temperature dependent specific heat.

Loading sources: constant and temperature dependent volume heat densities, convective and radiative sources, Joule heat sources imported from current flow analysis.

Boundary conditions: Prescribed temperatures, boundary heat flows, convection, radiation, and prescribed constraints for constant temperature boundaries.

Postprocessing results: temperatures, thermal gradients, heat flux densities, and total heat losses or gains on a given part; with transient analysis: graphs and tables of time dependency of any quantity in any given point of a region.

Special features: A postprocessing calculator is available for evaluating user-defined integrals on given curves and surfaces. Plate models with varying thickness can be used for thermal analysis. The temperatures can be used for thermal stress analysis (thermo-structural coupling). Special type of inter-problem link is provided to import temperature distribution from another problem as initial state for transient thermal analysis.

Stress Analysis

Stress analysis plays an important role in design of many different mechanical and electrical components. Generally the quantities of interest in stress analysis are displacements, strains and different components of stresses.

QuickField can perform linear stress analysis for 2-D plane stress, plane strain, and axisymmetric models. The program is based on Navier equations of elasticity. Following options are available for stress analysis:

Material properties: isotropic and orthotropic materials.

Loading sources: concentrated loads, body forces, pressure, thermal strains, and imported electric or magnetic forces from electrostatic or magnetostatic analysis.

Boundary conditions: prescribed displacements, elastic spring supports.

Postprocessing results: displacements, stress components, principal stresses, von Mises stress, Tresca, Mohr-Coulomb, Drucker-Prager, and Hill criteria.

CHAPTER 3

Problem Description

Structure of Problem Database

A special database is built for each problem solved with QuickField. The core of the database is *the problem description*, which is stored in file with the extension **.pbm**. The problem description contains the basics of the problem: its subject, plane, precision class, etc., and also references to all other files, which constitute the problem database. These files are the model file, with standard extension **.mod**, and physical data (property description) files, with extension **.dms**, **.dhe**, **.des**, **.dcf**, **.dht**, or **.dsa**, depending on the subject of the problem.

The problem description may refer to one or two files of physical data. Both files have the same format, and differ only in purpose. Usually, the first data file contains specific data related to the problem, as the second file is a library of standard material properties and boundary conditions, which are common for a whole class of problems.

Depending on the problem type, you may share a single model file or a single data file between several similar problems.

While solving the problem, QuickField creates one more file—the file of results with the extension **.res**. This file always has the same name as the problem description file, and is stored in the same folder.

Editing Problems

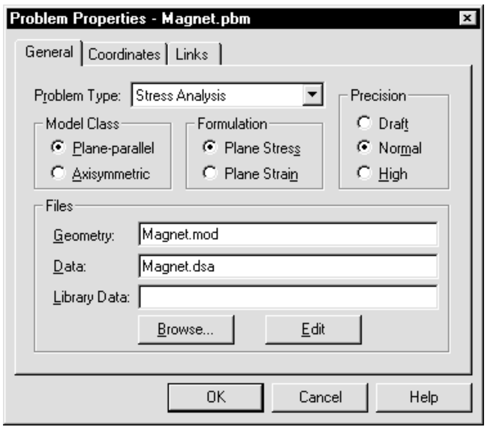
- To create a new, empty problem description, click **New** in the **File** menu and then select **QuickField problem** in the list that appears. Then enter the name and path of the new problem. You can also create a new problem as a copy of an another problem being currently opened. In that case new problem inherits all the properties of the sample one and the referenced model and data documents are copied if necessary.

- To open an existing document, click **Open** in the **File** menu, or use drag and drop features of Windows.

Open problem documents are shown in a special view to the left of main QuickField window. In problem view, you can edit problem description options and references to files. The tree shows the names of files, which the problem currently references.

- To change problem settings or file names, click **Properties** in the **Edit** menu or context (right mouse button) menu.
- To start editing a referenced document (model, data, secondary data or other problem referenced as coupling link), double-click its name in the tree, or click **Edit File** in the context menu, or click correspondent item in **Edit** menu.
- To solve the problem, click **Solve Problem** in the **Edit** menu or context (right mouse button) menu.
- To analyze the results, click **View Results** in the **Edit** menu or context menu.

Editing problem description properties



Problem type: Select the type of analysis, which your problem belongs to.

Model class: Select the geometry class of your model: plane or axisymmetric.

Precision: Select the precision you need. Note that higher precision leads to longer solution time.

Formulation: Select the formulation of planar stress analysis problem.

Frequency: Type the value of frequency for the time-harmonic magnetics problem. Note the difference between frequency *f* and angular frequency ω: $\omega = 2\pi f$.

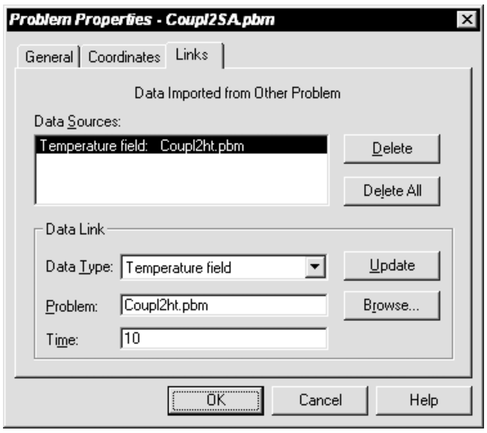
Files: Edit the file names of your model and data files. You may use long file names. If the name is given without the full path, it is assumed with respect to the problem description file. You can also click Browse to select file in any folder on your hard disk or the network.

Edit: Instantly loads selected file into the new QuickField window.

Establishing Coupling Links

The stress analysis and heat transfer problems can incorporate data, which come from other analysis types. The data types are: electrostatic and/or magnetic forces and temperature field for the stress analysis, and power losses generated by the current flow for the heat transfer. Transient heat transfer problems can import initial state of temperature distribution from another steady state or transient heat transfer problem (at specified time moment in latter case).

To establish a link between the problem that imports data and the problem that originates them, click **Links** tab in problem description dialog box.



To add a data link:

1. Select the type of the data in the **Data Type** list;
2. Type a name of the source problem in the **Problem** box, or click **Browse** button to make the selection from the list of existing problems;

3. In case the source problem is of transient analysis type, specify the time moment you wish to import in the **Time** field; if this specific time layer does not exist in the results file, the closest time layer will be imported;
4. And, click **Add** button to add the link to the list of data sources.

To change a data link:

1. Select the link of choice in the **Data Sources** list;
2. Change the source problem name or the moment of time as necessary;
3. And, choose **Update** button to update the link in the list of data sources.

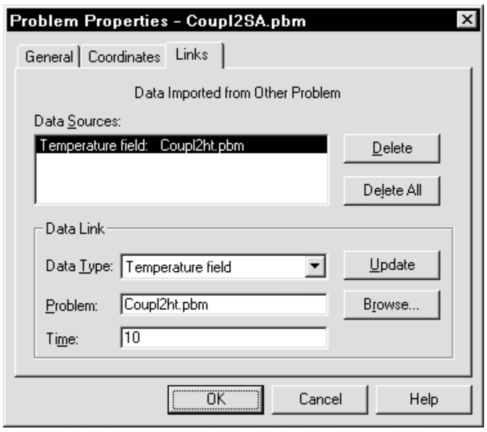
To delete a link:

1. Select the link of choice in the **Data Sources** list box;
2. And, click **Delete** button to delete the link from the list of data sources, or use **Delete All** button to delete all data links at once.

The links to the imported data are considered to be a part of the problem description. The changes made in them are preserved only if you choose **OK** when completing the problem description editing. And, vice versa, if you would choose **Cancel** button or press ESC, the changes made in data links will be discarded along with other changes in problem description.

Setting Time Parameters

With problems of transient analysis type, you need to set up the time parameters, before the problem can be solved. To do so, click **Timing** tab in the problem description dialog box.



Calculate up to: Specify the period of time you wish to simulate. Simulation always starts at ‘zero’ time moment.

With the step of: Specify the step size for the calculation. In transient analysis, this is the most important parameter controlling the precision of calculations in time domain: the smaller the step, the better the precision. Usually you will have minimum of 15 to 20 steps for the whole integration period. It may have sense to start with bigger value of this parameter and then decrease it if the result seems to change not smoothly enough.

If for some model you cannot estimate suitable time parameters, we recommend that you set some arbitrary value for the time period, and set the step size to have 5-7 points of integration, and then explore the X-Y plots against time in several points in the domain to tune the parameters.

Store the results every: defines the time increment for saving the results of calculation to the file. This value must be equal or greater than the step size.

Starting from the moment: defines the first point to be written to the file. If this value is zero, the initial state will be written.

Choosing Length Units

QuickField allows you to use various units for coordinates when creating model's geometry. You can use microns, millimeters, centimeters, meters, kilometers, inches, feet, or miles. To set the units of preference, choose **Coordinates** tab in problem description dialog box.

Chosen units are associated with each particular problem, which gives you freedom to use different units for different problems. Usually units of length are chosen before creating the model geometry. It is possible to change units of length later, but it does not affect physical dimensions of the model. So, if you create your geometry as a square with 1 m side and then switch to centimeters, you will get a square measured 100 cm by 100 cm, which is the same as it was before. To actually change size of the model you should rather use **Scaling** option of the **Move Selection** command of the Model Editor (see page 27 for details).

The choice of length units does not affect units for other physical parameters, which always use standard SI units. E.g., the current density is always measured in A/m^2 and never in A/mm^2. The only physical quantity that is measured in chosen units of length, is the displacement vector in stress analysis problems.

Cartesian vs. Polar Coordinates

Problem geometry as well as material properties and boundary conditions can be defined in Cartesian or polar coordinate systems. There are several places in QuickField where you can make choice between Cartesian and polar coordinate systems. Using **Coordinates** tab in problem description dialog box you can define the default coordinate system associated with a problem. The same option is also available in the Model Editor and in the Postprocessor. Definition of orthotropic material properties, some loads and boundary conditions depends on the choice of the coordinate system. You can choose Cartesian or polar coordinate system for each element of data individually and independently from the default coordinate system associated with the problem. This choice is available in the dialog boxes of the Data Editor.

CHAPTER 4

Model Geometry Definition

This chapter describes the process of building the *geometric model*—a type of QuickField document describing the problem geometry. It contains specific geometric objects and establishes the correspondence between the objects and material properties, field sources and boundary conditions

Terminology

Vertex, *edge* and *block* are three basic types of geometric objects, which constitute the model in QuickField.

Vertex is a point on the plane with coordinates defined by the user or calculated automatically as intersection of the edges. For each vertex you can define *the mesh spacing value* and *the label.* The mesh spacing value defines approximate distance between mesh nodes in the neighborhood of the vertex. The label is used, for example, to describe a line source or load.

Edge is a line segment or a circular arc connecting two vertices. It can't intersect any other edge of the region. If an edge being created contains an existing vertex, two adjacent edges are created. New vertices are automatically created in all points where new edge intersects the existing ones and all intersected edges are split by these vertices. Edges can be *labeled*, for example, to specify the boundary conditions.

Block is a continuous subregion with its boundary consisting of edges and possibly isolated vertices. A block may contain holes that can be formed by chains of edges or by isolated vertices. Each block has to be *labeled* to describe material properties. Labels of the blocks are also used to define distributed field sources. Unlabeled block is not included in calculation of field even it is covered by *the mesh*. The mesh is created block by block automatically or according to *the mesh spacing value* defined for particular vertices.

The Label is a string of up to 16-character length, which establishes the correspondence between geometrical parts of the model and physical values assigned to them. Any printable characters including letters, digits, punctuation marks, space character are permitted, except for asterisk (*) and question mark (?) characters. The label cannot begin with space character; trailing spaces are ignored. Labels are case-sensitive.

The Mesh Spacing value defines an approximate element size around the vertex. The mesh spacing parameter is associated with the vertex and measured in the current units of length. By setting mesh spacing values in some vertices you can control the mesh density and therefore the accuracy of the solution.

How to Create a Model

Model development consists of three stages:

- Geometry description;
- Definition of properties, field sources and boundary conditions;
- Mesh generation.

To describe model geometry you define vertices and edges, which form boundaries of all subregions having different physical properties. You can create vertices and edges; move, copy and delete any geometric object. To perform editing actions upon several objects at once, you can use *selection* mechanism.

You define properties, sources and boundary conditions by means of assigning labels to geometrical objects.

There are two options available for creating the finite element mesh for your model:

- Fully automated method which generates a smooth mesh with a density based on region's dimensions and sizes of geometrical details. This option does not require any information from the user.
- The second method allows you to choose the mesh density. In this case you need to define the spacing values at few vertices of your choice. Spacing values for other vertices are calculated automatically to make the mesh distribution smooth.

Creating Edges

To create new edges:

1. In the **Edit** menu or context (right mouse button) menu, click **Insert Mode** to switch the view into *insert mode*.
2. Use pull-down list box in the Model toolbar to choose new edge type (line segment or arc) and the arc angle—you may use predefined angles listed or type different value; zero angle corresponds to line segment.
3. Drag the mouse with left button pressed or use SHIFT+DIRECTION keys to drag the cursor from starting to ending point of the edge. You can use existing vertices as well as create new vertices while creating edges. If *snap to grid* feature is active, new vertices can be created only on grid nodes.
4. Please don't forget to switch the insert mode off after finishing insertion—otherwise you can easily insert unwanted objects!

Creating Vertices

To create new vertices:

1. In the **Edit** menu or context (right mouse button) menu, click **Insert Mode** to switch the view into *insert mode*.
2. Make sure that current *coordinate grid* settings fit coordinates of vertices you want to create.
3. Just use mouse or DIRECTION keys to move the cursor to point where you want new vertex to appear and double-click left mouse button or press ENTER.
4. Please don't forget to switch the insert mode off after finishing insertion—otherwise you can easily insert unwanted objects!

Or:

1. In the **Edit** menu, click **Add Vertices**.
2. Enter new vertex coordinates and click **Add**. Repeat if you need more vertices.
3. Click **Close**.

Objects Selection

To select geometric objects:

1. Make sure that insert mode is off.

2. Click objects you want to select with SHIFT or CTRL key pressed, or click and drag diagonally to select several connected objects at once. In latter case, only those objects are selected that entirely fit in the selection rectangle.

You also may use **Select All** and **Unselect All** commands in the **Edit** or context menu. Note that you can select objects of different types (blocks, edges or vertices) at once.

Copying and Moving Geometric Objects

Repeated geometry elements can be easily created by means of copying any set of objects to new location, using geometric transformations listed below. To make a copy:

1. Select any number of objects (vertices, edges and blocks) you want to copy, choosing **Select** from the menu.
2. In the **Edit** menu or context menu, click **Duplicate Selection**. The dialog box appears, asking for copying parameters.
3. Select transformation, enter its parameters and click **OK**. The new objects will appear on screen and the program will be waiting for your confirmation, so you could be sure that you entered the parameters correctly.
4. Click **Yes** to confirm copying. New objects will be ‘implanted’ into the model, and selection will move to the last copy.

The copy operation affects all explicitly set features of the selected objects, including labels and spacing values. Only the mesh is not copied.

Caution. Use copy operation with care, because improperly set transformation parameters may cause creating new objects in the wrong places. Such improper objects may interfere with the existing objects and generate a lot of useless intersection points, which will be hard to remove later.

You can also move selected objects to other location with the restriction that region topology will not change, and no new intersection or coinciding will arise. To move selected objects, click **Move Selection** in the **Edit** menu or context menu. The dialog box that appears is very similar to **Copy Selection** dialog box.

Geometric transformations available with move and copy operations are:

Displacement— parallel displacement is applied to selected objects for specified displacement vector. With copy operation, several copies can be asked for, it means that copying operation will be performed several times, each time being applied to the previous result. Parameters needed are displacement vector components.

Rotation — selected objects are rotated around the specified point for the specified angle. With copy operation, several copies can be asked for, it means that copying operation will be performed several times, each time being applied to the previous result. Parameters needed are center of rotation coordinates and angle measured in degrees.

Symmetry — selected objects are mirrored; symmetry line is specified by coordinates of any point on it and the angle between the horizontal axis and the symmetry line. Positive value of an angle means counter-clockwise direction. This transformation is available for copy operation only.

Scaling — selected objects are dilated (constricted) by means of homothetic transformation. Parameters needed are center of homothety and scaling factor. This transformation is available for move operation only.

Deleting Objects

To delete geometric objects:

1. Select objects you want to delete.
2. In the **Edit** menu or context menu, click **Delete Selection**.

If the selection contains vertices only and the vertex being removed connects exactly two edges, which can be treated as single edge when eliminating that vertex, those are joined together. Otherwise confirmation will be asked to delete all the connected edges.

Attraction Distance Parameter

To avoid small unrecognizable inaccuracies in geometry definition, new vertices or edges cannot be created very close to the existing ones. The creation of new geometric objects is controlled by the ε parameter also called the *attraction distance*.

The following rules concern creating new vertices and edges.

- Creating a new vertex is prohibited within 2ε-neighborhood of the existing one.
- A new edge cannot be added if it joins the same vertices as of an existing edge and the maximum gap between them does not exceed ε.
- If the distance between a vertex to add and some edge is less than or equal ε, the vertex is attracted by the edge and the edge is automatically split into pair of new edges to incorporate the vertex. The same is true when new edge is added, but in this case the new edge may be attracted by existing vertex.

The value of ε is 0.5 per cent of the visible region size, so to create very small details you have to zoom in the window.

Labeling Vertices, Edges and Blocks

The correspondence between geometrical objects and their physical properties, such as material properties, boundary conditions, or field sources is established by the use of labels.

To assign label:

1. Select objects you plan to give the same label
2. In the **Edit** menu or context menu, click **Properties**, the dialog box appears.
3. In **Label** list, type in the label or choose the label among existing ones. Then click **OK**.

If you select objects of different type at once, you can set labels to selected objects of each kind (blocks, or edges, or vertices) separately on different pages of Property dialog.

Meshing Technology

After creating the geometry of the model or its parts, you can proceed with building the finite element mesh. You can easily build a nonuniform mesh for a highly complex geometry. You may choose a fine mesh in some regions and very coarse in others, since the geometric decomposition technique would produce a smooth transition from large to small element sizes. Generally, the mesh has to be fine where the field changes most rapidly (high gradient), and also where you need high precision.

If the geometry is rather simple, or a draft precision for preliminary design analysis is satisfactory, it is suggested to use the fully automatic mode to create the mesh. With this option, once you built your geometry you would simply click **Build Mesh** and a suitable mesh is automatically created without any information on the mesh size.

You also have the option to pick the mesh density if you choose to do so. The mesh density is controlled by spacing values in vertices. The spacing value defines approximate distance between mesh nodes around that vertex. You never need to define the spacing in all model's vertices. To obtain uniform mesh you can set the spacing in any one vertex. This value is spread among all other vertices automatically. If you need the non-uniform mesh, define spacing values only in those vertices where you need finest and roughest mesh. The spacing values are automatically interpolated to other vertices to smooth the mesh density distribution. The group selection mechanism allows assigning the value to several vertices at once.

After defining spacing values, you can proceed with the mesh building. The mesh is built block by block. You may choose to build the mesh in one block or in selected blocks or in entire region at once.

Changing the density of a pre-built mesh (e.g. if solution results show that you need more precision somewhere in the region) obey some rules:

- When you change the spacing value in some vertex, the mesh is removed automatically in those blocks, which are connected to that vertex.

The mesh that is not removed, freezes spacing values along its boundary from recalculation as if those values were defined manually; so if you need major changes in the mesh density, first remove the mesh in the whole region.

To set mesh spacing:

1. Select vertices, edges or blocks, in neighborhood of which you need to specify the same spacing value.
2. In the **Edit** menu or context menu, click **Properties**.
3. Type in the **spacing** value or choose it from the list of already defined values, and then click **OK**.

If you choose to specify mesh spacing while selecting blocks or edges, the spacing value is actually assigned to all the vertices located on those edges or block boundaries.

To build the mesh:

- In the Edit menu or context (right mouse button) menu, click Build Mesh and then click appropriate option from submenu that appears.
- Or, select **Build Mesh** button on the toolbar. In this case the domain for building mesh is selected in following order:
  - In selected blocks, if any;
  - In labeled blocks, if any;
  - In all blocks in the region.

To remove the mesh:

- In the **Edit** menu or context (right mouse button) menu, click **Remove Mesh**. Then click appropriate option from submenu.
- Or, select Remove Mesh button on the toolbar. In this case the mesh is removed from selected blocks, if any, or from all meshed blocks.

If the spacing visibility switch is on (**Spacing** in **View** menu), the explicitly set spacing values are shown as small circles around the vertices. You can see the mesh building process if **Mesh** toggle in **View** menu is on.

Tuning the Picture in Model View

There are several options you can change to adjust the picture to best suit the task you are currently performing:

- **Scaling the picture (zoom)** gives you the ability to see more or less of your model to deal with small or large objects.
- Switching **visibility of model details** makes the picture more suitable to perform specific stage of model creating.
- **Background grid** makes the process of creating model vertices and edges easier and safer.

You also can open several windows for the same model and set different scaling factor and details visibility in each of them. To do so, click **New Window** in the **Window** menu.

Zooming

To magnify the picture:

1. Click **Zoom In** button in the model toolbar
2. Select the rectangle (click and drag diagonally), which then will occupy the whole window.

To see more of the model:

- Click **Zoom Out** button in the model toolbar.
- Or, click **Zoom to Fit** to see the whole model.

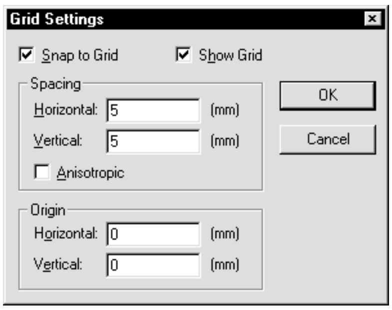
Model Discretization Visibility

There are four switches **Mesh, Domain, Breaking**, and **Spacing** which control the discretization visibility level. These are accessible in the **View** menu. When all these switches are off, region is displayed without discretization. This mode is useful for region geometry description and label setting. If the **Spacing** mode is switched on, all explicitly set spacing values are shown as circles with the appropriate radii.

When **Breaking** switch is on, the size of the elements along the edges is shown as tic marks on the edges. It is convenient to use both **Spacing** and **Breaking** when specifying the mesh spacing values. **Mesh** lets you see the complete triangular mesh. Turn it on to check the mesh building process. **Domain** without **Mesh** displays the domains due to geometric decomposition process.

Background Grid

Using the grid makes the process of creating model vertices and edges easier and helps to check the model. To change the grid, click **Grid Settings** in the **Edit** menu or context (right mouse button) menu.



Snap to Grid switches grid attraction on and off. Attraction means that mouse clicks create new vertices only at grid nodes making model description more fast and safe.

Show Grid switches on and off grid visibility.

Edit **Spacing** to change the grid density. To apply different horizontal and vertical spacing, first check the **Anisotropic** box.

Editing **Origin** gives you the ability to create vertices at even distances from certain point, which coordinates you enter here.

DXF File Import

You can import model geometry or its fragments from the DXF file produced by any major CAD system. To do so, choose **Import DXF** in the **File** menu and then type or select required file name. The visible region is automatically extended if needed to assure visibility of all imported geometric objects. If the model is not empty when reading the DXF file, it is recommended to save the current model’s state before the operation. This will give you a chance to return to the initial stage if the imported objects incidentally overlap the existing part of model.

DXF File Export

You can export model geometry or its fragments to the DXF file that can be read by any major CAD system or by QuickField itself . To do so, choose **Export DXF** in the **File** menu and then type or select required file name. If some geometrical objects in the model are selected, you can click appropriate button to choose whether to export the entire model or the selection only.

Printing the Model

You can directly print the model picture to your local or network printer, just as you see the model in the window, with the same zooming and discretization visibility.

- To print the picture, click **Print** in the **File** menu. You will have an option to choose the printer and set up the picture, such as paper size and orientation, before printing will occur.
- To preview the output before printing, click **Print Preview** in the **File** menu. To see how the picture will appear on a printer of your choice, click **Print Setup** before.

Copying the Model Picture

You can copy the model picture, as you see it in the window, to clipboard, for subsequent including it to your paper or report in any word-processing or desktop publishing utility.

- To copy the picture, click **Copy Picture** in the **Edit** menu, or press CTRL+INS.
- Switch to the application where you want to paste the picture and click **Paste** in the **Edit** menu, or press SHIFT+INS..

C H A P T E R 5

Problem Parameters Description

To solve the problem it is needed to describe the material properties, field sources and boundary conditions. These parameters are stored in the property description documents. The correspondence between records of these files and subdomains or boundaries of the region is established by the labels assigned to geometrical objects during editing the model. Labeling blocks, edges and vertices is described in Chapter 4 *"Model Geometry Definition"*.

The document consists of labels divided into three groups:

- block labels describe material properties and loads for subregions of the model;
- edge labels assign specific boundary conditions to your model's boundaries;
- vertex labels describe singular sources or constraints applied to points of your model.

Property description documents are specific to types of analysis. Each document occupies separate QuickField window and is stored in a separate disk file. File extensions are also specific to disciplines:

Kind of analysis	File extension
Magnetostatics	.dms
Time-harmonic magnetics	.dhe
Electrostatics	.des
Current flow	.dcf
Heat transfer	.dht
Stress analysis	.dsa

- To create a new property description, click **New** in the **File** menu and then select appropriate type of document in the list that appears.
- To open existing document, click **Open** in the **File** menu, or use drag and drop features of Windows, or, while working with problem description, double-click the name of associated property description file.

Editing material properties and boundary conditions

Once the property description document is opened, a new window appears in QuickField application window, displaying the structure of the document. The tree shows labels assigned to blocks, edges and vertices.

Icons displayed by the labels mean:

	Block label with specified material properties
	Edge label with specified boundary condition
	Vertex label with specified boundary condition or source
	Label referenced in models not yet given the properties
	Empty block label excluded from consideration
	Label with default boundary condition and zero source

Creating a New Label

To create a new label:

1. In the **Insert** menu, click **Block Label** or **Edge Label** or **Node Label**, or go to correspondent group of labels in the tree and click **New Label** in the context (right mouse button) menu.
2. New label will appear in the list prompting you to give it the name you want.
3. Just type the label's name you wish and press ENTER.

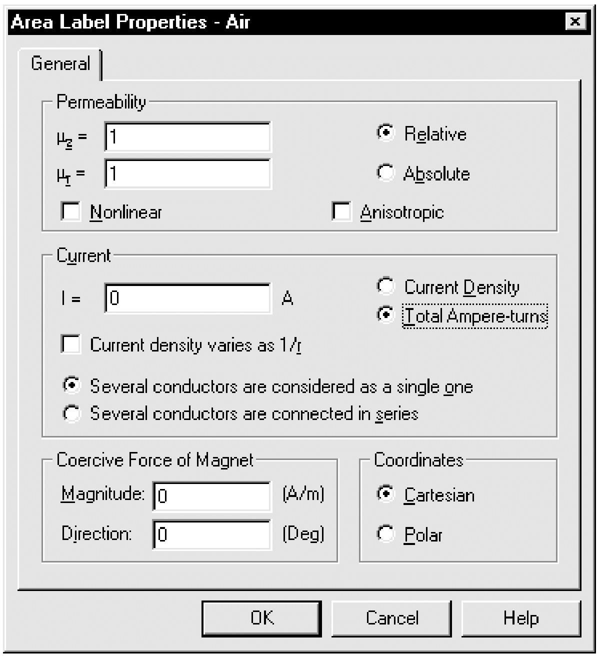
After you define the data, new label appears in the list of existing labels. If data editing was canceled, new label is not created.

Editing Label Data

To edit the data associated with some label, double-click the label in the list, or select the label and click **Properties** in the **Edit** menu or the context menu. The dialog box appears; its view depends on the class of current problem and on the type of geometrical object that the label corresponds to.

To finish label data editing, click **OK** button. Clicking **Cancel** button will end the editing and discards all changes to the values.

Editing Data in Magnetostatics



With problems of magnetostatics, block label data contain two components of magnetic permeability tensor, the current density, and for permanent magnets also magnitude and direction of coercive force.

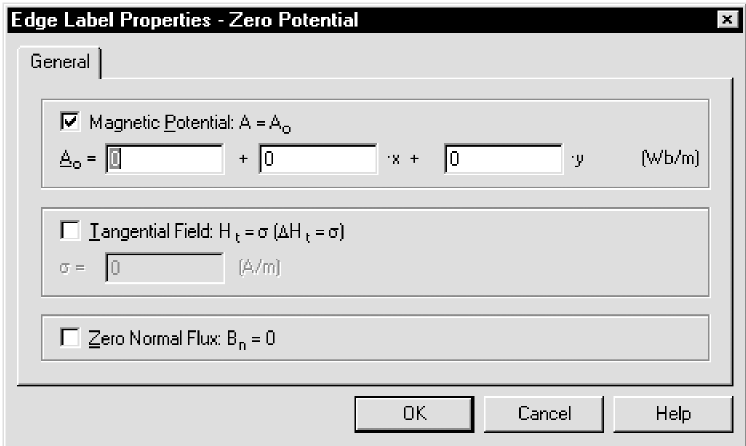
With nonlinear materials, you need to define the magnetization curve, instead of magnetic permeability. In this case check the **Nonlinear** box to get into the B-H curve editor. If a B-H curve had already been defined, the dialog box would contain a **B-H Curve** button that can be chosen to get into the curve editor. Editing the magnetization curve is discussed in *"Editing the Curves"* section later in this chapter.

When creating data for a new label, the text boxes for magnetic permeability components contain **None** instead of numbers. The word **None** in these boxes or the absence of the value means that the block with the corresponding label is excluded. If you want to define the material properties (and therefore include the block into consideration), simply type in a value of magnetic permeability, which will replace the highlighted **None**.

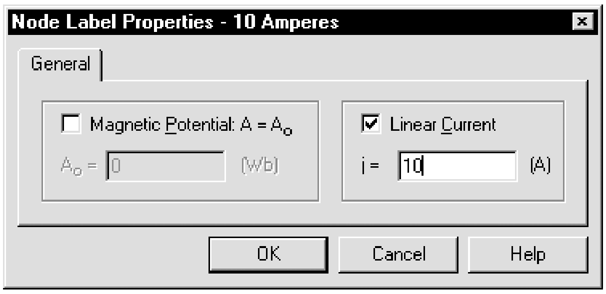
If you need to define two components different from each other, first check the **Anisotropic** box.

You can specify the kind of source (current densisty or total number of ampere-turns). If you have specified the total current, several blocks labeled with the same label can be considered as a single one or as conductors connected in series. Serial conductors are carring the same current and calculated current density could be different if their squares are not equal.

With axisymmetrical problems, if the total number of ampere-turns is specified you can define that current density in your coil varies inversely to the radius rather than being distributed uniformly. It might be closer to reality if your block represents a massive spiral coil.

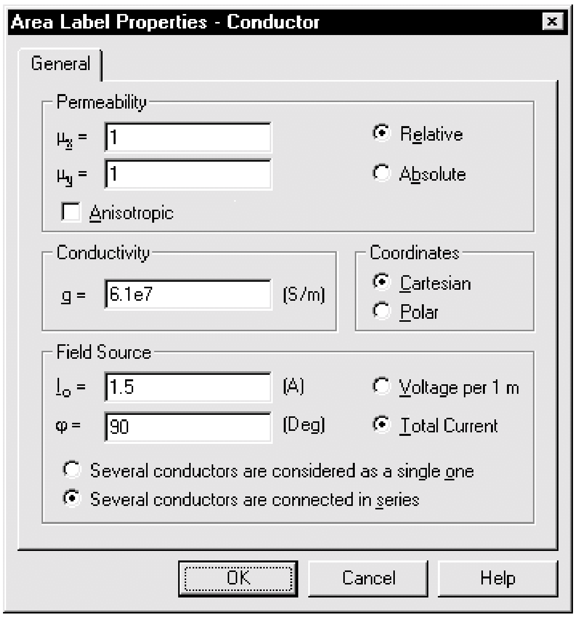


The data for the edge label allow to assign one of possible boundary conditions. Select the type of condition and then type in the values.



The vertex in the problem of magnetostatics may have known potential or the concentrated current may flow through the vertex. Check one of the options and then enter a value.

Editing Data in Time-Harmonic Magnetics



With problems of time-harmonic magnetics, block label data contain two components of magnetic permeability tensor, electric conductivity and one of three possible field sources: source current density, voltage, or total current.

When creating data for a new label, the text boxes for magnetic permeability components contain **None** instead of numbers. The word **None** in these boxes or the absence of the value means that the block with the corresponding label is excluded. If you want to define material properties (and therefore include the block into consideration), simply type in a value of magnetic permeability, which will replace the highlighted **None**.

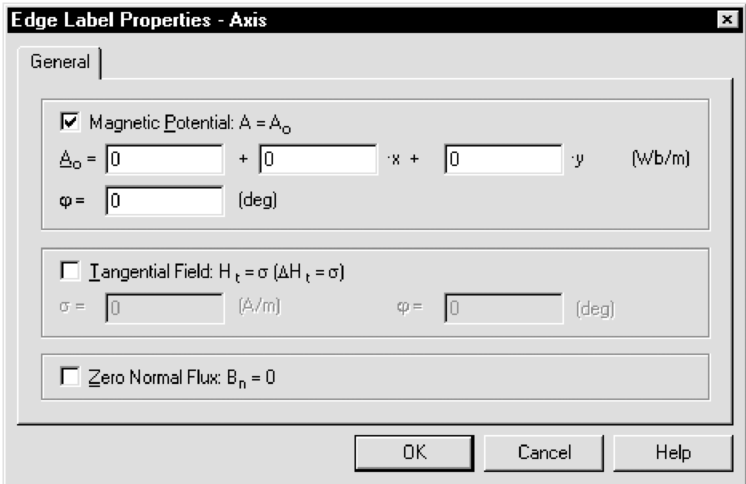
If you need to define the two components different from each other, first check the **Anisotropic** box.

The method of applying sources is different for conductors and non-conductive areas. In first case, you may switch between voltage and total current, as in second case voltage is inappropriate, and you can apply current density or total current only.

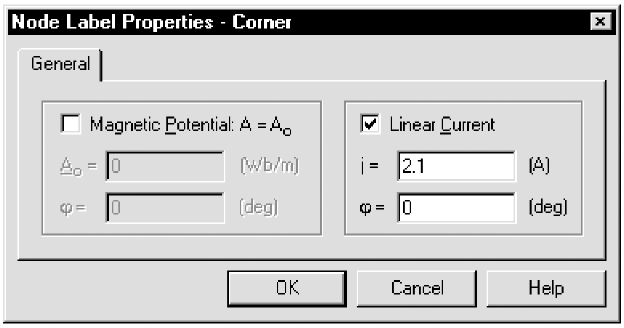
When total current or voltage is specified you can define several block labeled with this label (if any) as a single one or as different conductors connected in series. In the last case the total current over each conductor will be the same and distribution of the current density is subject to solve.

Note. It is assumed that the total current specified for a block label is the *gross* current in all blocks associated with that label.

With time-harmonic problems, you always specify amplitude, or peak, values for all alternating quantities.

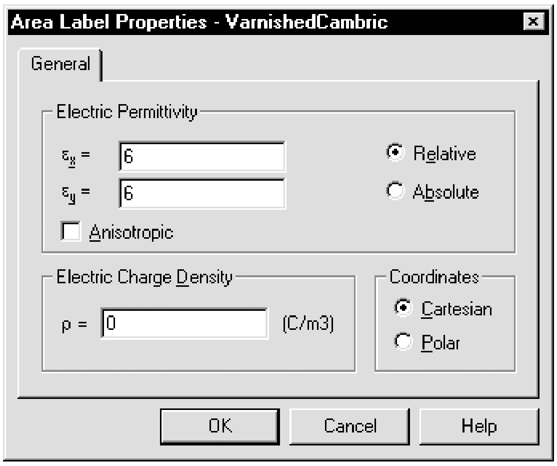


The data for the edge label allow to assign one of possible boundary conditions. Select the type of condition and then type in the values.



The vertex in the problem of time-harmonic magnetics may have known potential or a concentrated current may flow through the vertex. Check one of the options and then enter a value.

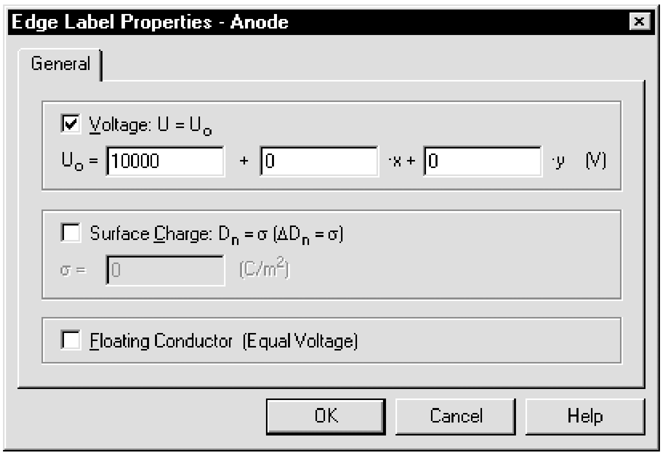
Editing Data in Electrostatics



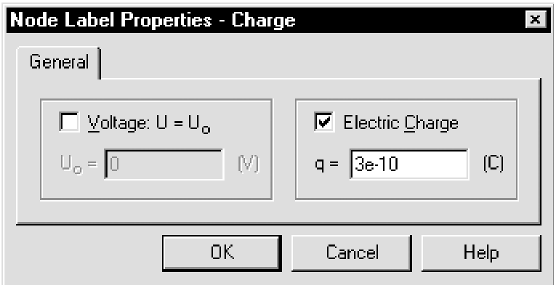
Block label data for electrostatics problem contain two components of electric permittivity and possibly distributed charge density.

When creating data for a new label, the text boxes for electric permittivity components contain **None** instead of numbers. The word **None** in these boxes or the absence of the value means that the block with the corresponding label is excluded. If you want to define the material properties (and therefore include the block into consideration), simply type in a value of electric permittivity, which will replace the highlighted **None**.

If you need to define two components different from each other, first check the **Anisotropic** box.

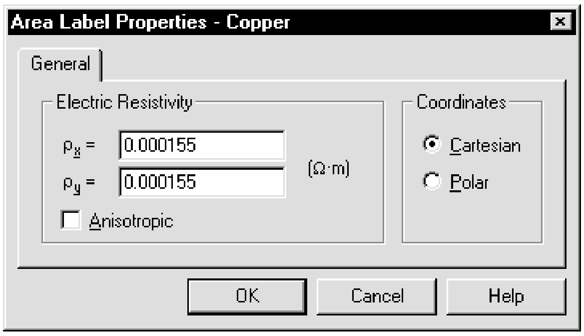


The data for the edge label allow to assign one of the possible boundary conditions. Select a type of condition and then type in the values.



The vertex in the problem of electrostatics may have known potential or concentrated charge. Check one of these options and then enter a value.

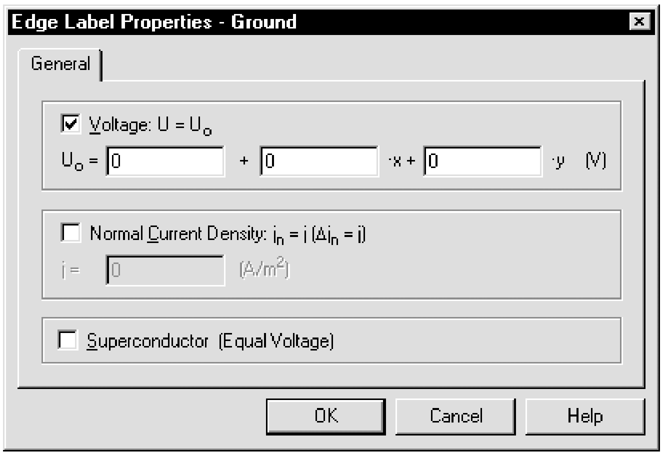
Editing Data with Current Flow Problems



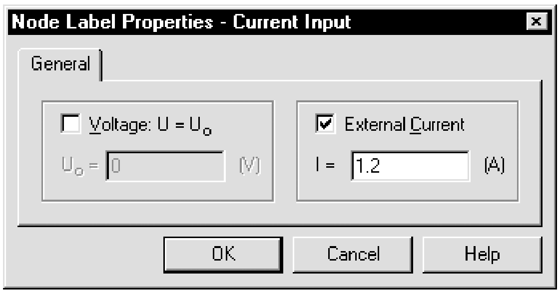
Block label data for the problem of current flow contain two components of electric resistivity.

When creating data for a new label, the text boxes for electric resistivity components contain **None** instead of numbers. The word **None** in these boxes or the absence of the value means that the block with the corresponding label is excluded. If you want to define material properties (and therefore include the block into consideration), simply type in a value of electric resistivity, which will replace the highlighted **None**.

If you need to define two different components of resistivity, first check the **Anisotropic** box.

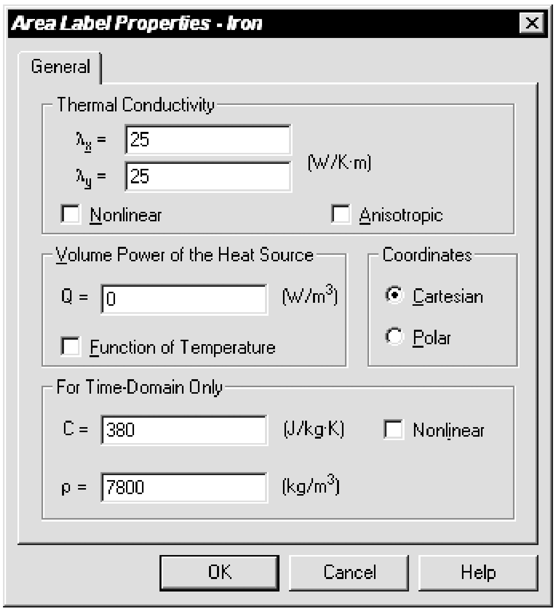


The data for the edge label allow you to assign one of possible boundary conditions. Select the type of condition and then type in the values.



The vertex in the problem of current flow may have known potential or external current. Check one of these options and then enter a value.

Editing Data with Heat Transfer Problems



The data for block label contain two components of thermal conductivity tensor and, possibly, the volume power of heat source. For transient analysis, the values of specific heat and volume density are also required.

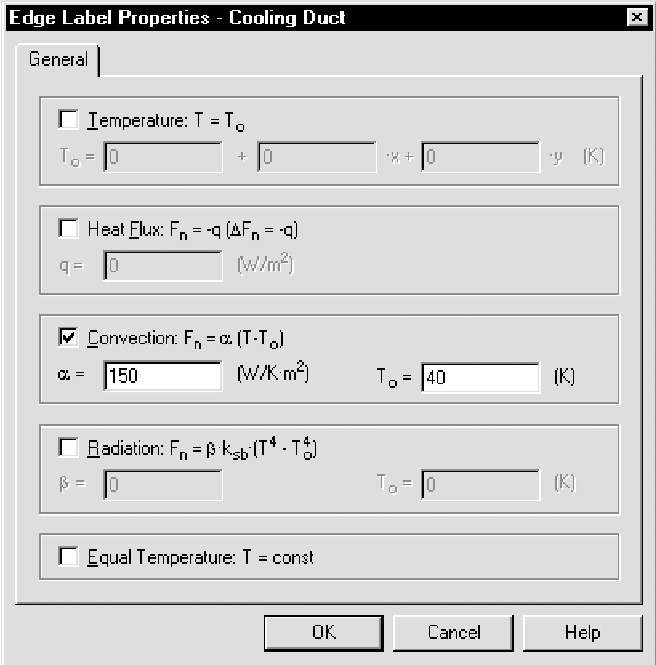
To describe the thermal conductivity as a function of temperature, check the **Nonlinear** box and the temperature curve editor for defining $\lambda = \lambda(T)$ will be displayed. Curve editing is discussed in *"Editing the Curves"* section later in this chapter.

Also the volume power of heat source could be described as a function of temperature. To do so, check the **Function of Temperature** box related to the heat source field. Editing the dependencies is described in *"Editing the Curves"*.

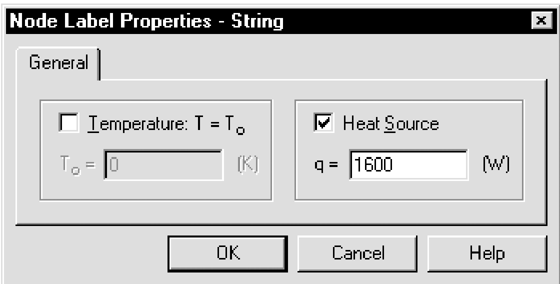
Specific heat *C* can be specified as either the constant value or as a function of temperature. In latter case, check the **Nonlinear** box to bring up the Curve Editor for specific heat.

When creating new label, the text boxes for thermal conductivity components contain **None** instead of numbers. The word **None** in these boxes or the absence of the value means that the block with the corresponding label is excluded. If you want to define material properties (and therefore include the block into consideration), simply type in a value of thermal conductivity, which will replace the highlighted **None**.

If you need to define two different components of thermal conductivity, first check the **Anisotropic** box.



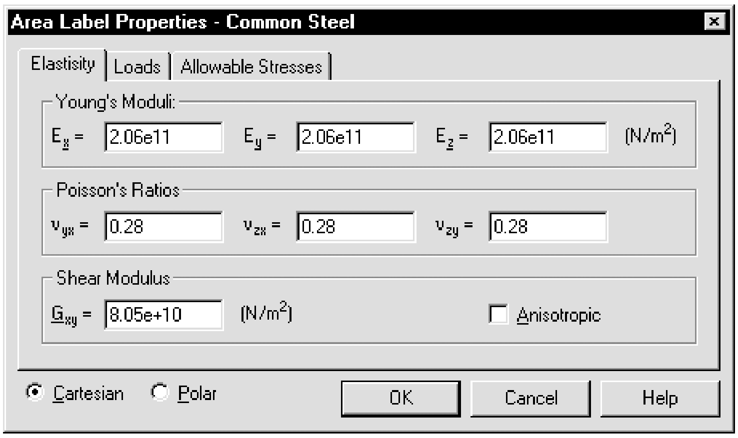
The data for edge label allow you to describe boundary conditions. Check the condition that you need, and then type in the parameters. The heat flux, convection, and radiation can be combined together, which means that the heat flow through the surface is compounded from several components.

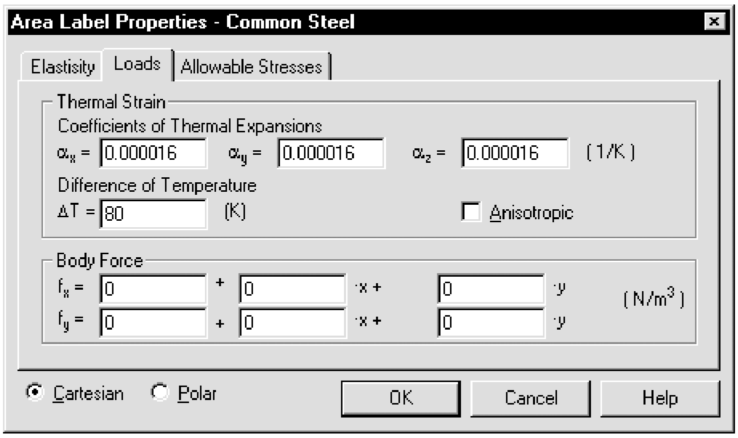


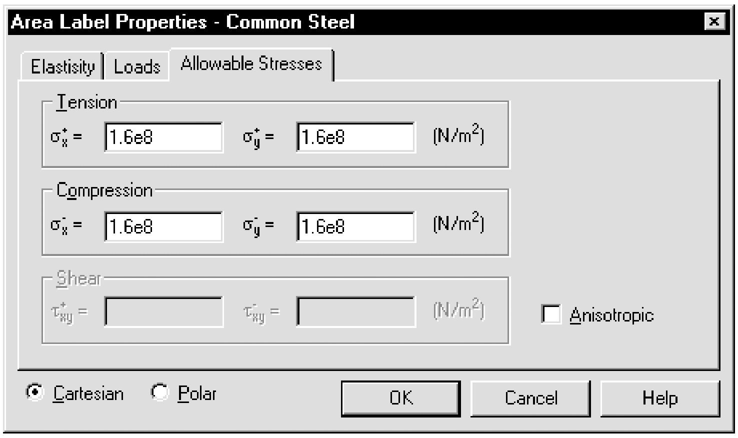
The vertex in heat transfer problem may have known temperature, or represent a line heat source. Check one of these options, and then enter the numeric parameter.

Editing Data with Stress Analysis Problems

The data for the block label with stress analysis problem are spread between three tabs in a dialog box.







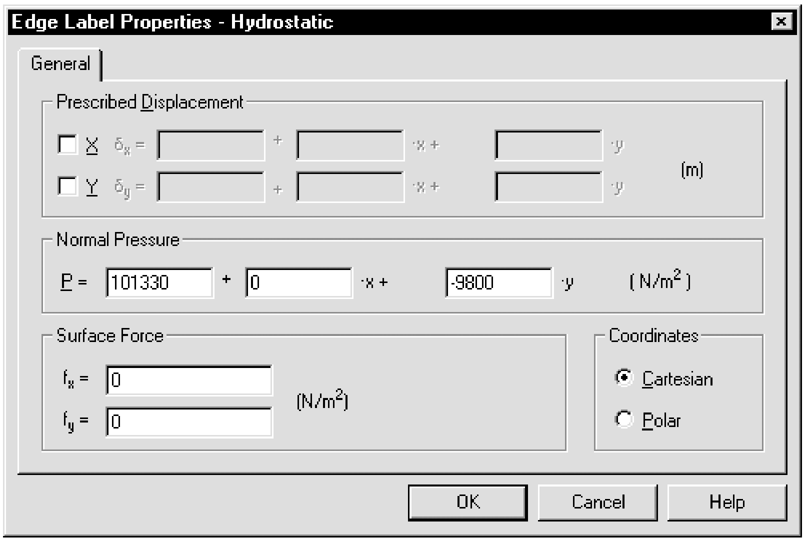
When creating new label, the text boxes for Young's moduli contain **None** instead of numbers. The word **None** in these boxes or the absence of the value means that the block with the corresponding label is excluded. If you want to define material properties (and therefore include the block into consideration), simply type in a value of the Young's modulus, which will replace the highlighted **None**.

The **Anisotropic** boxes, which apply to elastic moduli and coefficients of thermal expansion, allow you to describe anisotropic properties in each set independently.

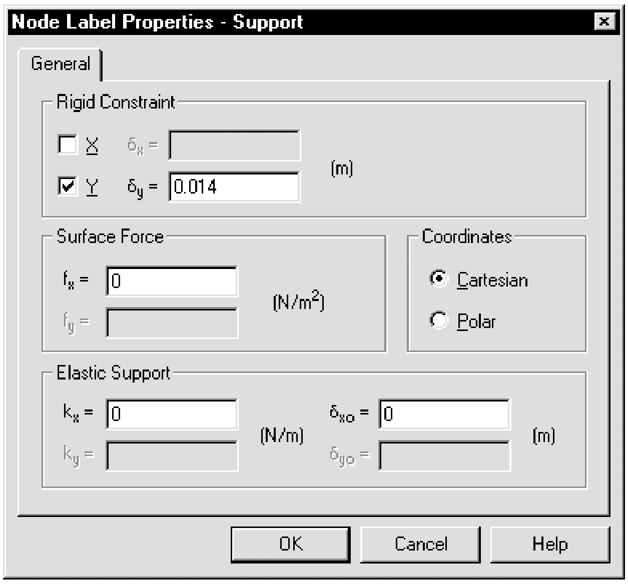
The data for thermal loading are defined slightly different way for thermo-structural coupled and non coupled problems:

- With an uncoupled problem, you define the temperature difference between strained and strainless states, which is assumed to be constant within all blocks with the corresponding label.
- With thermo-structural coupling, you need to define a reference temperature of strain free state for each block subjected to thermal loading.

The values of allowable stresses do not affect the solution. Those are only used in postprocessing stage to calculate the Mohr-Coulomb, Drucker-Prager, and Hill criteria. You don't need to define allowable stresses, if these criteria are of no interest to you.



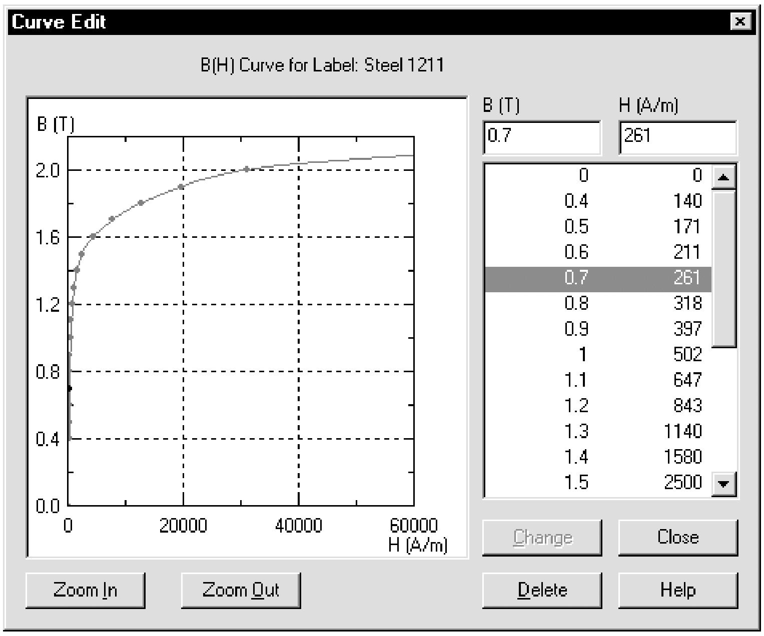
The data defined for an edge label may include constraints along one or both coordinate axes and the surface forces are described either as normal pressure or by their Cartesian or polar coordinate system components. To apply fixed displacement along an axis, check the appropriate box and then enter a value of displacement.



The vertex label data may define rigid or elastic support along one or both coordinate axes, or concentrated external force. To describe rigid constraint along some axis, check the appropriate box, and then enter the value of fixed displacement.

Editing the Curves

Curve functions, which describe some field dependent parameters, are implemented as tables containing two columns: an argument and a function, e.g., magnetic field intensity and flux density or temperature and thermal conductivity. Editing the table is supported with graphical presentation of the dependency, which is interpolated with cubic spline between the entered points. The solver uses just the same curve as you see on your screen.



To add the new point to the dependency, type in two values (*B* and *H* in shown example) and press ENTER key or choose **Add** button. If the argument of a new point coincides with the argument of existing one, new point replaces the old one.

To remove the point, select it in the table and choose **Delete** button or press the DEL key.

You may control the scaling of the graph with use of **Zoom In** or **Zoom Out** buttons, or simply by clicking and dragging diagonally in the graph.

To exit from editing the curve, choose **Close** button or press ESC. Note that subsequent canceling of label data editing with ESC key or the **Cancel** button will discard all changes including the curve editing.

Copying, Renaming and Deleting Labels

Labels can be copied within single property description document or between documents of the same type.

To copy the label:

1. In the list, select the label you want to copy with right mouse button and click **Copy** in the context menu.
2. Switch to destination window and click **Paste** in the **Edit** menu or context menu.

Or,

1. Drag the label to destination position with the mouse.

To delete the label:

- In the list, select the label with right mouse button and click **Delete** in the context menu.
- Or, select the label and click **Delete** in the **Edit** menu.

To move (cut and paste) the label:

1. In the list, select the label you want to move with right mouse button and click **Cut** in the context menu.
2. Switch to destination window and click **Paste** in the **Edit** menu or context menu.

Or,

1. Drag the label to destination position with the mouse holding down the SHIFT key.

CHAPTER 6

Solving the Problem

This chapter describes how to solve the prepared problem, and methods QuickField uses to solve.

Several conditions have to be met to solve a problem. The problem type, plane, required precision and other parameters have to be specified in the problem description file. The model geometry file must contain complete model with mesh and labels. Each label referred by the model file is to be defined in the problem's private or library data file.

To obtain the problem solution, click **Solve Problem** in the **Edit** menu or context (right mouse button) menu of the Problem editor. You may skip this action and directly proceed to the analysis results by clicking **Analyze Results** in the **Edit** or context menu. If the problem has not been solved yet, or its results are out of date, the solver will be invoked automatically.

Each solver runs in its separate thread, so you can solve several problems at once or edit or analyze other problems while the problem is being solved. There is of course no sence in editing any document related to the problem being solved.

Special bar indicator lets you see the progress of the solution process. Linear problems are solved by using a powerful preconditioned conjugate gradient method. The preconditioning based on the geometric decomposition technique guaranties a very high speed and close to linear dependence between number of nodes and the resulting solution time. Nonlinear problems are solved using the Newton-Raphson method. The Jacobian matrix arising at each step of the Newton-Raphson method is inverted the same way as it is done for linear problems.

We use the Euler's method (constant time step size) for solving transient problems, with initial value set to zero or taken from another temperature field calculation. This method is extremely fast and stable, however we recommend having at least 15-20 time steps for whole transitional process to achieve accurate and smooth results.

Achieving Maximum Performance

The algorithm used in QuickField solver does not require the whole data of the problem to fit into memory of your computer. The solver can effectively handle linear algebraic systems with matrices several times bigger than the amount of available physical memory. Data that don’t fit into memory are stored on the hard disk. The size of the problem you can solve on your computer is only limited by the amount of free disk space. Memory consumption is very low compared to other FEA packages, only about 1.3 MB per ten thousand degrees of freedom.

Although size of the problem is not limited by the amount of available memory, having additional memory may improve performance. It is obvious that the performance is the best when all the data can be stored in memory and relatively slow disk access is not used during solution.

However, to solve very large problems on a computer with insufficient memory it is essential that virtual memory is configured optimally.

To manage virtual memory settings:

1. Bring up Control Panel and double-click **System**.
2. Switch to **Performance** tab.
3. See Windows Help for details.

C H A P T E R 7

Analyzing Solution

This chapter explains the procedures for detailed examination of the results using the QuickField postprocessing utility.

To analyze the problem solution, choose **View Results** in the **Edit** menu or context menu of the problem window. The Postprocessor provides various ways of results' presentation:

- field pictures,
- local field values,
- integral quantities,
- X-Y plots,
- tables.
- tables and plots vs. time for transient problems.

Any picture or numerical value displayed by the postprocessor can be copied to Windows clipboard for use with any word-processing or desktop publishing utility or subsequent use by spreadsheet or user-written programs.

Building the Field Picture on the Screen

Interpreted Quantities

The set of the physical quantities, which can be displayed by the Postprocessor, depends on the problem type.

For the electrostatic problem these quantities are:

- Scalar electric potential (voltage) U;
- Vector of electric field intensity $\mathbf{E} = -\,\mathbf{grad}U$;

- Tensor of gradient of electric field G **= gradE**;
- Vector of electrostatic induction **D =** ε**E**;
- Electric permittivity ε (or its largest component in anisotropic media);
- Electrostatic field energy density $w = (\mathbf{E} \cdot \mathbf{D})/2$.

For the magnetostatic problem:

- Vector magnetic potential A in plane-parallel problem or flux function $\Phi = 2\pi rA$ in axisymmetric case;
- Vector of magnetic flux density **B = curl A**;
- Vector of magnetic field intensity $\mathbf{H} = \mu^{-1} \cdot \mathbf{B}$;
- Magnetic permeability μ (its largest component in anisotropic media);
- Magnetic field energy density:

$w = (\mathbf{B} \cdot \mathbf{H})/2$ —in linear media,

$w = \int (\mathbf{H} \cdot \mathbf{dB})$ —in ferromagnetic media.

For the time-harmonic electromagnetic problem:

- Complex amplitude of vector magnetic potential A (flux function rA in axisymmetric case);
- Complex amplitude of voltage U applied to the conductor;
- Complex amplitude of total current density $j = j_0 + j_{\text{eddy}}$, source current density j_0 and eddy current density $j_{\text{eddy}} = -i\omega gA$.

  All these complex quantities may be shown in form of momentary, root mean square (RMS) or peak value in time dimension.

  E.g., complex quantity $z = z_0 e^{i(\omega t + \phi_z)}$ my be shown as:

  - momentary value at a given phase $\phi_0 = \omega t_0$

    $$z_{\phi_0} = \mathrm{Re}\left[z_0 e^{i(\phi_0 + \phi_z)}\right] = z_0 \cos(\phi_0 + \phi_z) ;$$

  - peak value z_0;
  - RMS value

    $$z_{\text{RMS}} = \frac{\sqrt{2}}{2} z_0 .$$

- Complex vector of the magnetic flux density **B = curl A**

$$B_x = \frac{\partial A}{\partial y}, \qquad B_y = -\frac{\partial A}{\partial x}$$ —for planar case;

$$B_z = \frac{1}{r}\frac{\partial(rA)}{\partial r}, \qquad B_r = -\frac{\partial A}{\partial z}$$ —for axisymmetric case;

- Complex vector of magnetic field intensity $\mathbf{H} = \mu^{-1}\mathbf{B}$, where μ is the magnetic permeability tensor.

  Complex vectors may be shown in form of momentary, RMS or peak magnitude.

- Time average and peak Joule heat density $Q = g^{-1} j^2$;
- Time average and peak magnetic field energy density $w = (\mathbf{B} \cdot \mathbf{H})/2$;
- Time average Poynting vector (local power flow) $\mathbf{S} = \mathbf{E} \times \mathbf{H}$;
- Time average Lorentz force density vector $\mathbf{F} = \mathbf{j} \times \mathbf{B}$;
- Magnetic permeability μ (its largest component in anisotropic media);
- Electric conductivity g.

For the problem of current flow:

- Scalar electric potential U;
- Vector of electric field intensity $\mathbf{E} = -\,\mathbf{grad}U$;
- Vector of current density $\mathbf{j} = \rho^{-1}\cdot\mathbf{E}$;
- Electric resistivity ρ (its largest component in anisotropic media);
- Ohmic losses per volume unit $w = (\mathbf{j} \cdot \mathbf{E})/2$.

For heat transfer problem:

- Temperature T;
- Vector of heat flow $\mathbf{F} = -\lambda\cdot\mathbf{grad}(T)$;
- Thermal conductivity λ (its largest component in anisotropic media).

For stress analysis problem:

- Displacement vector δ;
- Strain tensor ε and its principal values;
- Stress tensor σ and its principal values;
- Von Mises stress (stored energy of deformation criterion):

  $$\sigma_e = \sqrt{\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]};$$

  where σ_1, σ_2 and σ_3 denote the principal stresses in descending order.

- Tresca criterion (maximum shear):

  $$\sigma_e = \sigma_1 - \sigma_3;$$

- Mohr-Coulomb criterion:

$$\sigma_e = \sigma_1 - \chi\sigma_3 ,$$

where

$$\chi = \frac{[\sigma_+]}{[\sigma_-]} ,$$

$[\sigma_+]$ and $[\sigma_-]$ denote tensile and compressive allowable stress.

- Drucker-Prager criterion:

$$\sigma_e = \left(1+\sqrt{\chi}\right)\sigma_i - \frac{\sqrt{\chi}-\chi}{1+\sqrt{\chi}}\bar{\sigma} + \frac{1}{[\sigma_-]}\left(\frac{1-\sqrt{\chi}}{1+\sqrt{\chi}}\bar{\sigma}\right)^2 ,$$

where

$$\sigma_i = \sqrt{\frac{1}{2}\left[(\sigma_1-\sigma_2)^2 + (\sigma_2-\sigma_3)^2 + (\sigma_3-\sigma_1)^2\right]} ;$$

$$\bar{\sigma} = \frac{\sigma_1+\sigma_2+\sigma_3}{3} .$$

- Hill failure index for orthotropic materials:

$$F.I. = \frac{\sigma_1^2}{X_1^2} - \frac{\sigma_1\sigma_2}{X_1^2} + \frac{\sigma_2^2}{X_2^2} + \frac{\tau_{12}^2}{S_{12}^2} ,$$

where σ_1, σ_2 and τ_{12} are computed stresses in the material directions and,

$X_1 = X_1^T$ if $\sigma_1 > 0$; $X_1 = X_1^C$ if $\sigma_1 < 0$

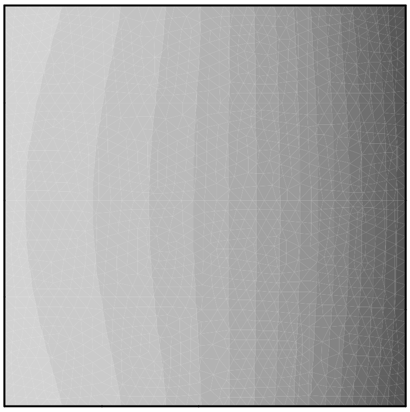
$X_2 = X_2^T$ if $\sigma_2 > 0$; $X_2 = X_2^C$ if $\sigma_2 < 0$

$S_{12} = S_{12}^+$ if $\tau_{12} > 0$; $S_{12} = S_{12}^-$ if $\tau_{12} < 0$

The Hill failure index is calculated only for those materials, where allowable stresses were defined (while editing the block data, see *"Problem Parameters Description"*). If any pair of allowable stresses is not given, the corresponding term is dropped while calculating the Hill Index.

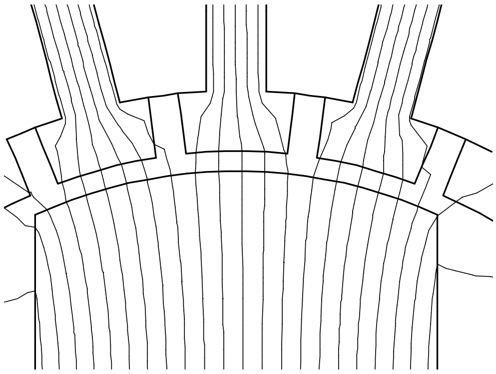
Field Presentation Methods

Several methods are available for displaying the field picture:



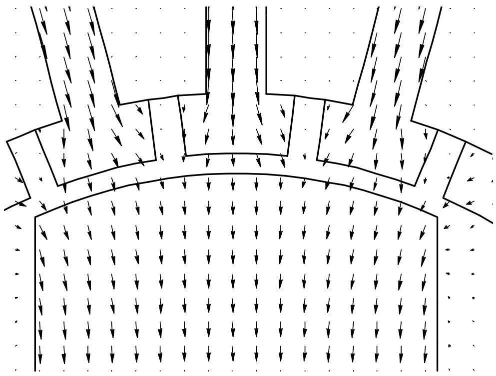
Color map for distribution of a chosen scalar quantity. The color map is accompanied by the legend showing the correspondence between colors and numerical values.

You can adjust the color scale by changing the range limits for the chosen quantity.
Color map may be shown in gray scale mode if you want to optimize it for monochrome printing.



Field lines. Those are isotherms for temperature fields, lines of equal potential in electrostatics and flux lines for magnetostatic problems.

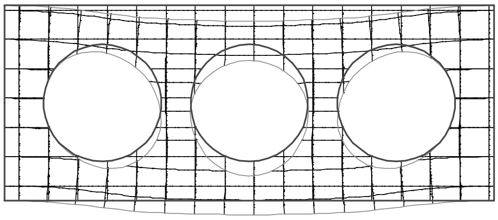
You can manipulate the picture by changing the distance between neighboring lines. This distance is measured in units of chosen quantity.



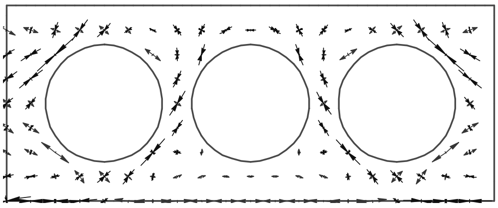
Vectors—family of line segments showing magnitude and direction of the vector quantity. Vectors are drawn in the nodes of the regular rectangular grid.

You can change the grid cell size and the scaling factor for a desired vector quantity.

The following methods are specifically for stress analysis problems:



Deformed boundary and shape indicated by means of deformed and original rectangular grid.



Stress tensor display as a pair of eigenvectors reflecting the direction of principal axes, magnitudes and signs of principal stresses (blue color denotes tension, red color—compression);

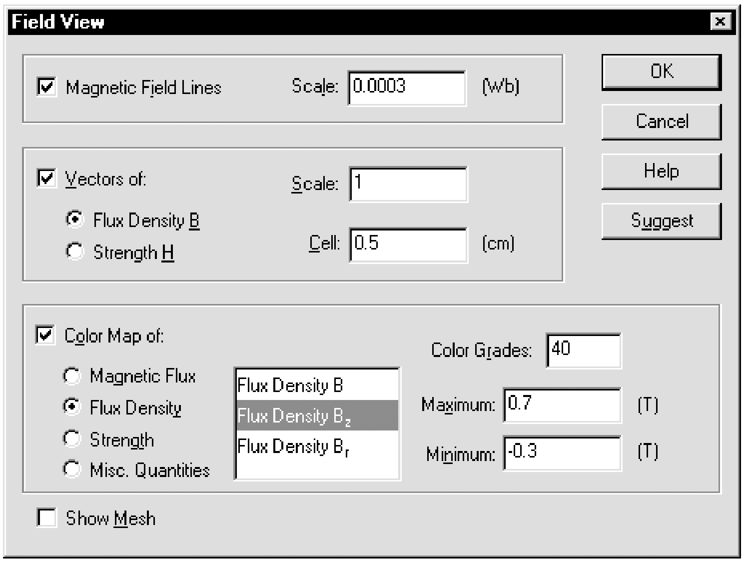
With these methods, you can change the grid cell size and the scaling factors in order to manipulate the appearance.

It is possible to combine several visualization methods in the same picture to obtain the most expressive result.

QuickField can display several different field pictures for the same problem. To open a new window, click **New Window** in the **Window** menu.

Field Picture Constructing

When entering the Postprocessor, the default form of the field picture appears on the screen. You may use **Field Picture** in the **View** menu or context menu to select other display methods or quantities.

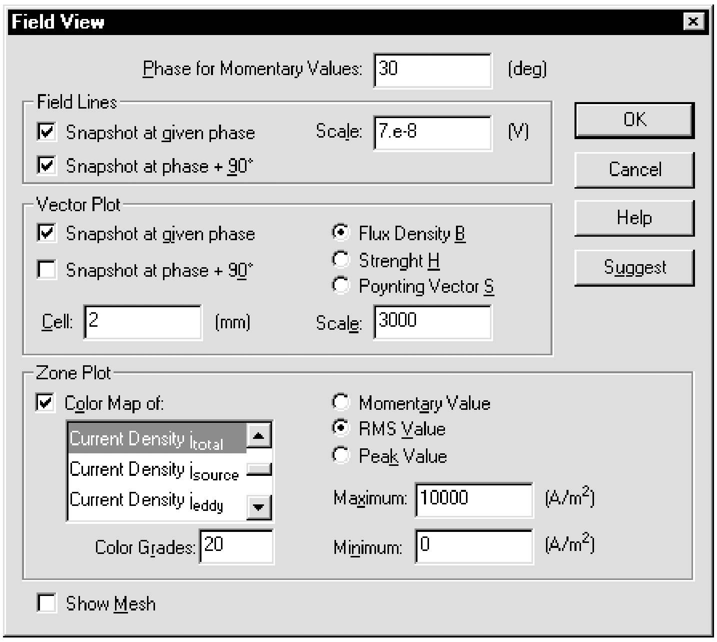


Shown dialog box corresponds to the problem of magnetics.

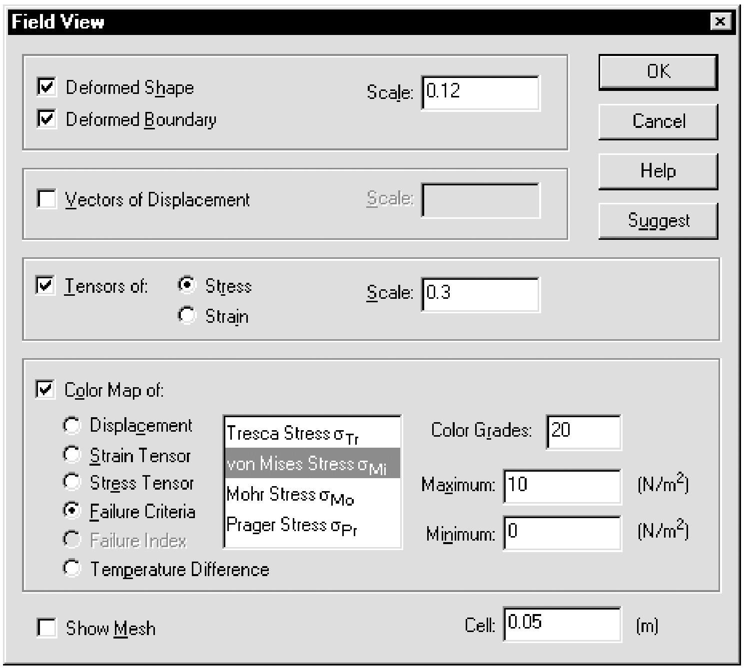
To choose desired visualization method, select corresponding check box. You can select any combination of methods at once. If none of the methods is selected, only the model's geometry is shown.

This dialog box also allows changing scaling parameters for selected methods of presentation and the number of color grades used with the color map. When you select some edit box, you can choose **Suggest** button to obtain suggested value of corresponding parameter. Note that suggested values for **Minimum** and **Maximum** fields are calculated for the currently visible part of the model.

In case of time-harmonic electromagnetic problem, equilines and vectors are drawn at specified phase. The **Field View** dialog box allows setting phase value. For more expressive field picture, you can order the second family of equilines or vectors, shifted with regard to the first by 90°.



The **Field View** dialog box for the stress analysis problem additionally allows to select tensor quantity visualization.



Sizes of the vector symbols for all vector quantities except the displacement vector are determined by the corresponding physical value multiplied by the scaling factor and by the cell size. Similar method is used for stress tensor components. Unlike other vector quantities, the size of the displacement vector on the screen does not depend on the cell size. It is determined by the dimensionless scaling factor, the unit value of which means that the displacement is shown in its natural scale.

Color map of temperature difference in stress analysis problem visualizes temperature distribution as it is defined by user or imported from linked heat transfer problem. In the last case, temperature is shown only in those blocks, where it is really taken into account.

The **Failure Index** option is available when the model contains at least one block with correctly defined allowable stresses.

Choosing the **OK** button causes redrawing the field picture on the screen. **Cancel** closes the dialog box without redrawing the picture and preserves preceding values of all the parameters.

Zooming

Zooming in postprocessor view is very similar to the analogous option of the Model Editor.

To magnify the picture:

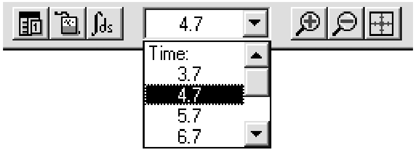
1. Click **Zoom In** button in the toolbar
2. Select the rectangle (click and drag diagonally), which then will occupy the whole window.

To see more of the model:

- Click **Zoom Out** button in the toolbar.
- Or, click **Zoom to Fit** to see the whole model.

Selecting a Time Layer

With a transient problem a field picture displayed corresponds to the specific time moment. A Time combo-box in the postprocessor toolbar displays the currently selected time layer and allows you to change it. Initially the very first time layer is displayed.



The field pictures, XY-Plots and tables redraw automatically when you change the time value, but the scaling factors set for several displaying methods do not change.

Calculator Window

Calculator Window is a window normally docked to the left side of the field view.

To open the calculator window, choose **Calculator Window** command in the **View** menu or corresponded button on the postprocessor toolbar. The calculator window also opens when choosing **Local Values**, **Integral Values** or one of the **Wizard** commands in **View** menu.

The calculator window is organized in several trees which root items correspond to several kinds of numerical data. These are:

- **Local Values** shows several field quantities at a point of interest;
- **Integral calculator** lists available quantities calculated by integration over given line, surface or volume;
- **Inductance Wizard** opens wizard, which helps you calculate self or mutual inductance of the coils and conductors,
- **Capacitance Wizard** opens wizard guiding you through steps needed to calculate self or mutual capacitance of your conductors in electrostatics problems,
- **Impedance Wizard** opens wizard, which helps you calculate the impedance of the conductors in AC magnetics problems.

To open the set of values, double-click the corresponding item, or select it and press ENTER.

The calculator window is initially docked to the left side of the field view. To change its width, point to the gray splitter strip between windows and drag it to the left or to the right. You can dock the window to the right side of the field view or make it floating as ordinary popup window. Point at the window caption and drag it to the desired position.

You can select one or several items in the tree and copy them to the clipboard or drag to any application that supports drag-and-drop copy/paste operation (almost any word processor or spreadsheet). To select more than one item, click on it holding the SHIFT key (block selection) or the CTRL key (random selection). Context (right mouse button) menu also works in the calculator window. It provides you with the subset of commands for manipulating the field picture in the active view.

With a transient problem all the values in the calculator window correspond to the chosen time layer. See Selecting a Time Layer"*Selecting a Time Layer*" section above for more details.

Examining Local Field Data

To obtain local field data, click **Local Values** in the **View** menu or context (right mouse button) menu in field picture window. Otherwise if the calculator window is already open, double-click the **Local Values** item in the tree. The message appears prompting you to click the point. Then you can click points where you need to know the values of the field quantities.

To enter coordinates of the point of interest from keyboard, select any of coordinates with mouse and then click it again (after a period, to avoid the double-click effect) or choose **Edit Point** command from context menu. You can edit either Cartesian or polar coordinates.

To leave this mode, close local values window, or choose **Local Values** in the menu again or click corresponding button on the toolbar.

The local values of physical quantities obtained in the **Local Values** mode can be copied to clipboard for printing numerical results, or to pass them to other application program, e.g., a spreadsheet program to produce a report. Click the **Copy** button in the Local Values window. To see or copy exactly those field quantities you need, you can expand or collapse branches in the tree.

Parameter Calculation Wizards

The most common design parameters in QuickField are calculated through wizards. These calculations still could be done by using the ordinary integral quantities available in the postprocessor, but wizards allow you to get the results quicker and in most cases you can avoid manual building of the contour of integration and manipulating with complex values.

These three wizards are available in QuickField:

- Inductance Wizard calculates self or mutual inductance of the coil or conductor in AC or DC magnetics problems,
- Capacitance Wizard calculates self or mutual capacitance of the conductors in electrostatics problems,
- Impedance Wizard calculates impedance of the conductor in AC magnetics problems.

To start the wizard, choose **Wizard** in **View** menu, or double-click the corresponding item in the calculator window. If the calculator window is open while you start the wizard, all the parameters calculated by the wizard are shown in that view. You can start wizard again from not only its start page but also from any other page by double-clicking the corresponding value in the Values tree.

Some of the wizards provide several alternative ways to calculate the desired quantity. Each way is represented in the calculator window as a separate tree.

Inductance Wizard

Inductance wizard helps you to calculate self and mutual inductance of your coils in the problem of magnetostatics or time-harmonic magnetics.

When your model contains several coils that carry different currents, the flux linkage with one of them can be calculated as

$$\phi_k = L_{kk} i_k + \sum_n M_{nk} i_n ,$$

where L_{kk} is the self inductance of the coil k, M_{nk} is the mutual inductance between the coils n and k, i_k is the current in the coil k.

On the other hand, stored magnetic energy also derives from current and inductance:

$$W = \frac{1}{2}\left(\sum_k L_{kk} i_k^2 + \sum_{n \neq k} M_{nk} i_n i_k \right)$$

Before using the inductance wizard, you have to formulate your problem in such a way that all the currents (space, surface or linearly distributed) but one are set to zero. There must be no permanent magnets in your model. In that case equation above becomes extremely simple and you can get inductance value as:

$$L = \frac{\phi}{i},$$

where ϕ is the flux linkage with the coil excited by current i, or

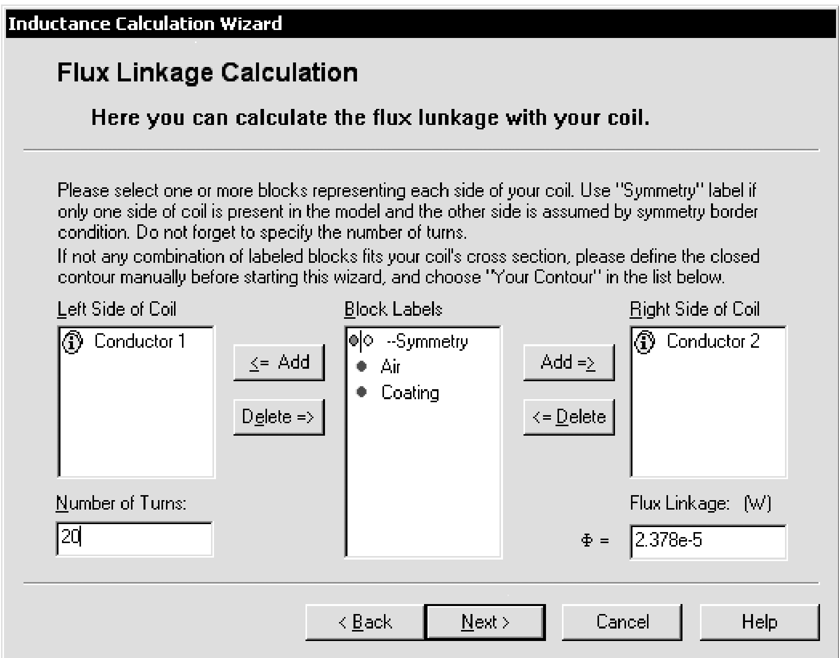
$$L = 2 \cdot \frac{W}{i^2},$$

where W is stored magnetic energy and i is the only current.

The first approach gives the self-inductance, if you get the flux linkage and the current in the same coil and mutual inductance if the coils are different. The second approach gives only the self-inductance.

Initial page of the inductance wizards invites you to choose between two approaches described above. After choosing one of them click the **Next** button.

The second page of inductance wizard allows you to define, which blocks represent the cross section of your coil. In general, two blocks represent each coil in the model plane: forward and return wires. If there is only one side of the coil in your model, the second one is assumed as being symmetrical to the first one or as being infinitely distant of the model and not affecting the field distribution.



To define each side of your coil, simply point the corresponding item in the **Block Labels** list and drag it to one of the side list. You can also use the **Add** buttons. No matter, which side of your coil you call **Right Side** and which **Left Side**. If only one side of the coil is represented in the model, drag item **Symmetry** to the opposite list if return wire of the coil is symmetrical to the direct one, or leave the list empty if return wire does not affect the electromagnetic state of your model.

You can select and drag more than one item at once if the cross section of your coil is split to several blocks.

Enter the **Number of Turns** for your coil if it is more than one.

As result of any action on the lists or number of turns the **Flux Linkage** value will change automatically being calculated as

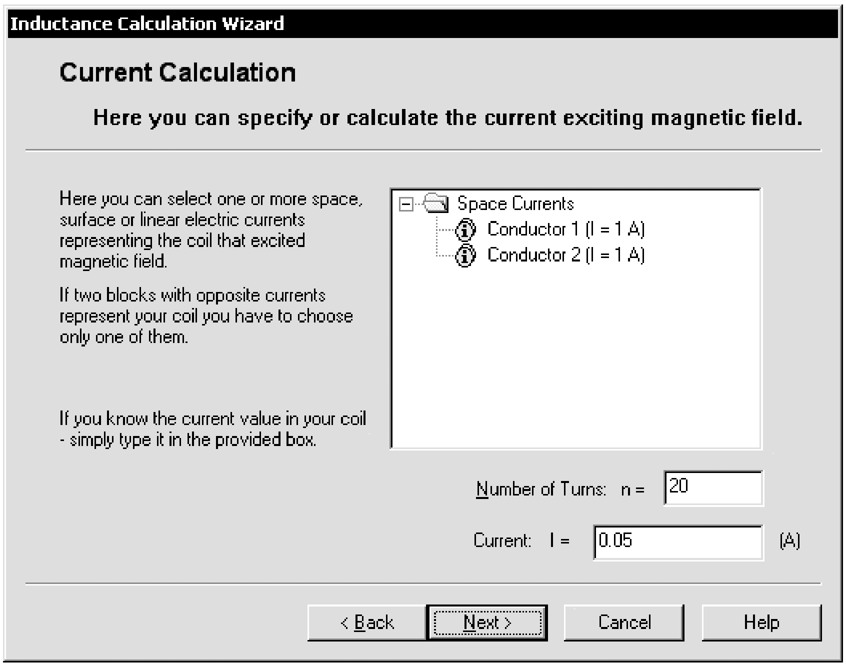
$$\phi = N \cdot \left(\frac{\int_L A \cdot ds}{\int_L ds} - \frac{\int_R A \cdot ds}{\int_R ds} \right)$$ for planar case

$$\phi = 2\pi \cdot N \cdot \left(\frac{\int_R Ar \cdot ds}{\int_R ds} - \frac{\int_L Ar \cdot ds}{\int_L ds} \right)$$ for axisymmetric case

where *A* is the vector magnetic potential; R and L denote the right and the left side of the coil accordingly, r is the radius of the point.

For planar problems flux linkage and the inductance are calculated per one meter of axial depth no matter what length unit you have chosen.

When you finish with flux linkage calculation, click on the **Next** button. In the **Current** page you can select the current exciting the field and provide a number of turns in your coil.



Capacitance Wizard

Capacitance wizard helps you to calculate self and mutual capacitance of your conductors.

When your model contains several conductors, the charge of one of them can be calculated as:

$$W = \frac{1}{2}\left(\sum_{k} C_{kk}U_k^2 + \sum_{n\neq k} C_{nk}U_nU_k\right),$$

where C_{kk} is the self capacitance of the conductor k, C_{nk} is the mutual capacitance between the conductors n and k, U_k is the voltage drop on the conductor k.

On the other hand stored energy also derives from charge and capacitance as:

$$W = \frac{1}{2}\left(\sum_{k} C_{kk}U_k^2 + \sum_{n\neq k} C_{nk}U_nU_k\right),$$

and from the voltage and capacitance as:

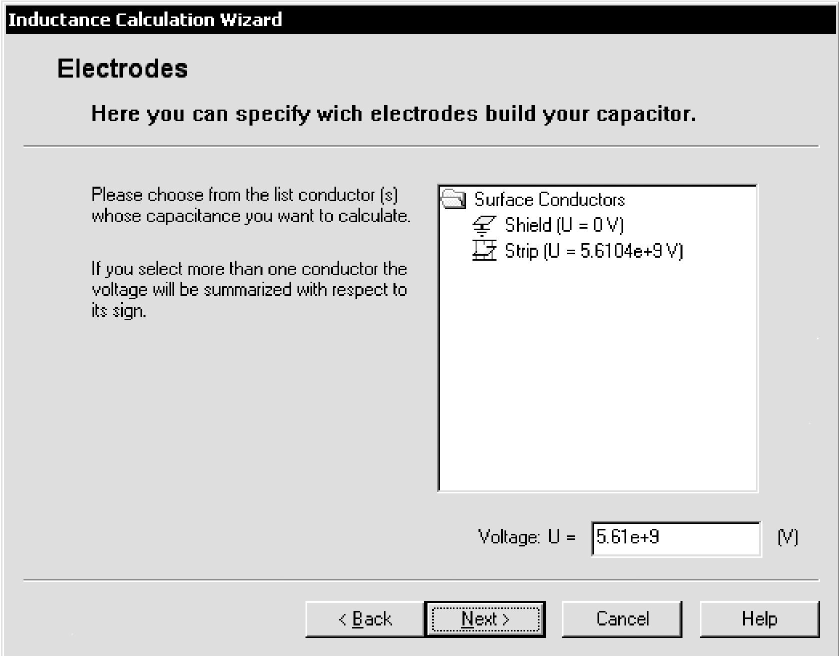
$$W = \frac{1}{2}\left(\sum_k \frac{q_k^2}{C_{kk}} + \sum_{n \neq k} \frac{q_n q_k}{C_{nk}}\right)$$

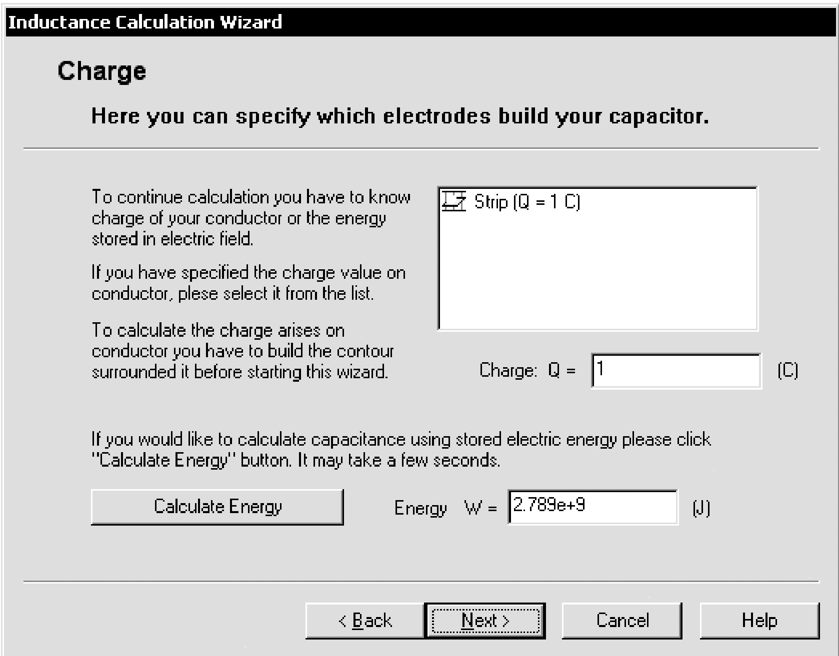
Before using the capacitance wizard, you have to formulate your problem in such a way that all field sources (space, surface or linear distributed charge or voltage) but one are set to zero. In that case equation above becomes extremely simple and you can get capacitance value if you know any two of these three quantities: charge, voltage, stored energy

When formulating your problem, you can apply known voltage to the conductor and measure the charge it produce or vice versa. Measuring the charge is a bit more complex than the voltage. It requires you to build the closed contour surrounding your conductor (but not coinciding with its surface) before you start the capacitance wizard. The easiest way to formulate the problem for capacitance calculating is to put constant potential boundary condition on the conductor's surface and specify an arbitrary non zero electric charge in one of the vertices on the surface of the conductor.

This page of capacitance wizard allows you to specify electrodes which capacitance you want to calculate. Electrodes listed on the right side of the page are organized in two subtrees: surface conductors and linear conductors (if any).

In case you are calculating the capacitance of the condenser consisting of two electrodes, select both of them. When choosing more than one electrode their voltage will be sum up (with their sign).


Inductance Calculation Wizard
Electrodes
Here you can specify wich electrodes build your capacitor.
Please choose from the list conductor (s) whose capacitance you want to calculate.
If you select more than one conductor the voltage will be summarized with respect to its sign.
Surface Conductors
Shield (U = 0 V)
Strip (U = 5.6104e+9 V)
Voltage: U =
5.61e+9
(V)
< Back
Next >
Cancel
Help


Inductance Calculation Wizard
Charge
Here you can specify which electrodes build your capacitor.
To continue calculation you have to know charge of your conductor or the energy stored in electric field.
If you have specified the charge value on conductor, plese select it from the list.
To calculate the charge arises on conductor you have to build the contour surrounded it before starting this wizard.
Strip (Q = 1 C)
Charge: Q =
1
(C)
If you would like to calculate capacitance using stored electric energy please click "Calculate Energy" button. It may take a few seconds.
Calculate Energy
Energy W =
2.789e+9
(J)
< Back
Next >
Cancel
Help

In the right side of the page electrodes are listed which charge you have specified. If you have put voltage boundary condition rather then charge on your electrodes, the only way to calculate the charge is to build the contour surrounding it but not coincident with it boundary. If so, you have to do that before you start the capacitance wizard.

When selecting one or more items in the list, you get the resulting charge in the Charge box.

Impedance Wizard

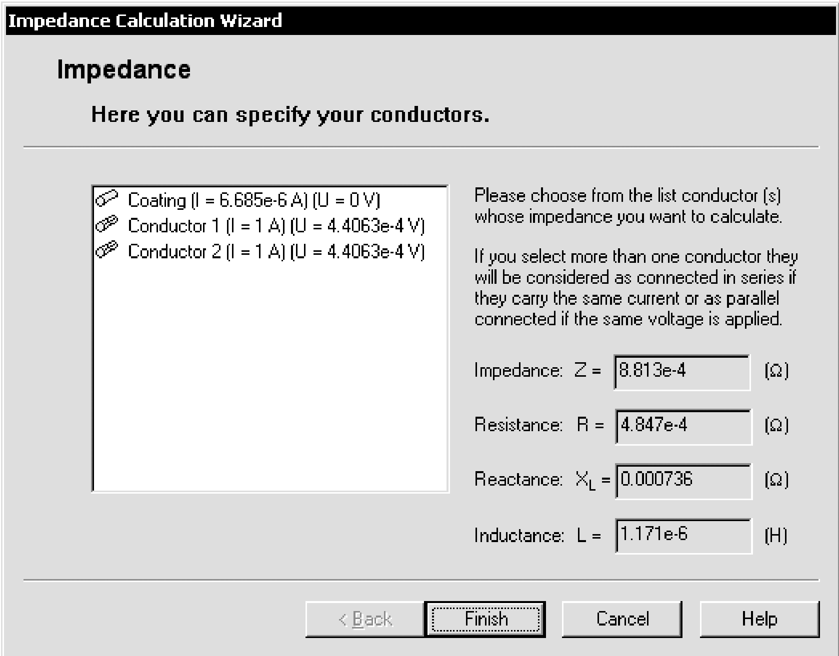
Impedance wizard helps you to calculate the impedance of your conductors. It is simple and contains only one page. To get the impedance value and its real and imaginary parts (resistance and reactance accordingly) the impedance wizard simply divides complex values of voltage by current:

$$Z = \frac{U}{I}$$

$$R = re(Z), X_L = im(Z);$$

$$L = \frac{X_L}{2\pi f},$$

where Z is absolute value of the impedance and f is the frequency.



If you select more than one conductor at once, the impedance wizard considers it as being connected in parallel if the voltage applied to each of them is equal and as being connected in series otherwise.

Editing Contours

The contour is a directed curved line consisting of line segments and arcs (including the edges of the model). Some rules are applied to the contours:

- The contour may not intersect itself.
- Open and closed contours are discerned.
- Multiply connected contours have sense only for calculating integral quantities.

Contour is shown in the field picture window as a set of directed lines or color-filled interior (closed counter-clockwise-directed contours).

QuickField allows editing contours in field picture windows. The following operations change the current contour state:

Adding lines	attaches a line segment or an arc to the contour. The arc is specified by its degree measure (zero means line segment)

and two end points. Any arbitrary line may initiate the contour, but only adjacent lines are accepted later. The line cannot be added to the closed contour. There are two ways to add lines to contour: choose **Pick Elements** from **Contour** menu or context menu and then drag mouse with left button pressed. Or, choose **Add Lines** from **Contour** menu or context menu and enter end points coordinates from keyboard.

Adding edges	appends the contour with an edge of the model. The contour may be initiated by any arbitrary edge, but only adjacent edges are accepted later. The edge cannot be added to the closed contour, or if the ending point of the contour does not currently coincide with model's vertex. To add edges, choose **Pick Elements** from **Contour** menu or context menu and then pick series of adjacent edges with mouse.
Adding blocks	considers the current closed contour as a border of the plane region and updates that region by adding (or subtracting) a block of the model in the sense of set theory. To add blocks, choose **Pick Elements** from **Contour** menu or context menu and then pick blocks with mouse.
Close contour	closes an open contour by connecting its open ends with a straight line or an arc, depending on current degree measure in the postprocessing toolbar.
Change direction	alters the contour direction.
Clear	deletes the entire contour.
Delete last	deletes the last element (line or edge) in the contour. Not applicable to multiply connected contours.

Depending on current state of the contour, some editing operations may be prohibited.

The direction of the contour is significant in the following cases:

- For volume integrals, the domain of integration lies to the left of the contour.
- For surface integrals, the positive normal vector points to the right relative to the contour direction.

- The starting point of the contour corresponds to zero point at the *x*-axis of the X-Y plot.
- If the plotted or the integrated function has different values to the left and to the right of the contour, the right-hand value is used.

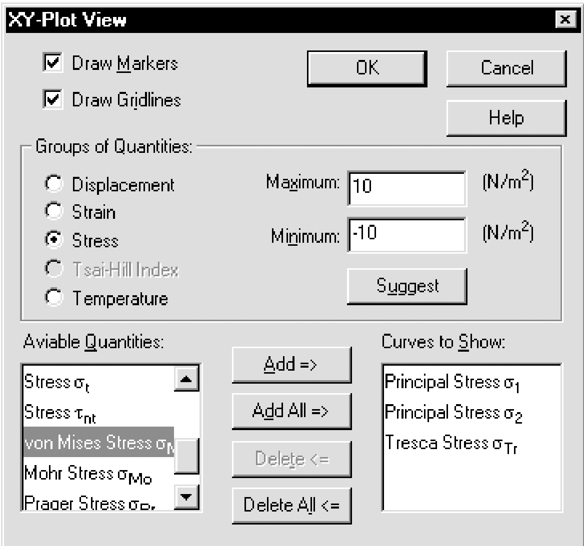
X-Y Plots

QuickField postprocessor can display field distribution along contours. To open new X-Y plot window, choose **X-Y plot** in **View** menu or context (right mouse button) menu in field picture window, in which the contour is already defined.

In X-Y plot view, you can:

- Select the set of shown quantities. Click **X-Y Plot Curves** in the **View** or context menu.
- Zoom the plot in or out.
- View the correspondence between quantities and curves (legend).
- Copy the picture to clipboard.
- Open new X-Y plot window for the same contour.

X-Y Plot Control



Few quantities having the same unit of measurement can be shown at the same X-Y plot. According to this, all quantities are combined into groups. Full list of quantities includes all those available for the color map representation (see *"Interpreted Quantities"*), and also normal and tangential components of vector and scalar quantities.

When you select the appropriate group of quantities, the **Curves to Show** list contains the quantities selected for display, and the **Available Quantities** list contains available but not selected quantities. You can use buttons located between the lists, or simply double-click in the lists, to move some quantity from one list to another.

In the dialog box, you can also modify the range of *y* coordinate. By default, it fits all the currently selected curves. You can get the suggested value of lower or upper limit by selecting the corresponding text box (**Minimum** or **Maximum**) and choosing **Suggest** button.

In time-harmonic analysis, you can also switch between momentary (at given phase), time average and peak values of time dependent quantities.

You can turn on or off the switches for displaying coordinate grid and markers on the curves. The last mode allows you to distinguish between the coinciding curves.

Calculating Integrals

QuickField calculates line, surface and volume integrals. In plane-parallel problem, a contour defines cylindrical (in generalized sense) surface of infinite depth, or volume of that cylinder for volume integral. Therefore, *in plane-parallel formulation surface and volume integrals are calculated per unit depth.* In axisymmetric problem, a contour defines toroidal surface, or toroid for volume integral.

Positive direction of a contour is counter-clockwise. The direction of the contour is accounted as follows:

- For volume integrals the domain of integration lies to the left of the contour.
- For surface integrals the positive normal vector points to the right relative to the contour direction.
- If the plotted or the integrated function has different values to the left and to the right of the contour, the right-hand value is used.

Force, torque and electric charge integrals represent real physical quantities only when the contour is closed. However, these integrals are calculated for the unclosed contours too.

To calculate integrals, click **Integral Values** in the **View** menu or context (right mouse button) menu. Or, if calculator window is already open, double-click on the **Integral Calculator** item in the tree. If the contour is already defined, a list of available integral quantities appears. The list varies depends on whether your contour is open or closed. If you have no contour defined in the active field view, a message appears prompting you to build the contour. You can get a value of an integral parameter by click on the small gray button left on its name or by double click on the name. Once opened the integral value will be recalculated automatically each time you change the contour.

Some integrals require closed counter-clockwise oriented contour, otherwise they have no physical sense. Once you created the contour, you can select an integral quantity from the list and choose **Calculate** button to get the value. **Copy** button allows you to copy the calculated result to clipboard.

When the electrostatic or magnetic force, torque, electric charge, electric current or heat flux are to be calculated, the domain of integration may be chosen by many different ways. The only requirement for the surface of integration is to contain all the necessary bodies, but to avoid any extra bodies or field sources. It is important to understand that the accuracy will be the best if you choose the integration surface as far as possible from the places with strong inhomogeneity of field, e.g., field sources or boundaries of conducting or ferromagnetic bodies.

When calculating the flux linkage the domain of integration must exactly fit the cross section of the coil.

The quantities available for electrostatic problems:

- Total electric charge in a particular volume

  $$q = \oint \mathbf{D} \cdot \mathbf{n} ds \ ,$$

  where integral is evaluated over the boundary of the volume, and **n** denotes the vector of the outward unit normal.

- Total electrostatic force acting on bodies contained in a particular volume

  $$\mathbf{F} = \frac{1}{2} \oint \left(\mathbf{E}(\mathbf{n} \cdot \mathbf{D}) + \mathbf{D}(\mathbf{n} \cdot \mathbf{E}) - \mathbf{n}(\mathbf{E} \cdot \mathbf{D}) \right) ds$$

- Total torque of electrostatic forces acting on bodies contained in a particular volume

  $$\mathbf{T} = \frac{1}{2} \oint \left((\mathbf{r} \times \mathbf{E})(\mathbf{n} \cdot \mathbf{D}) + (\mathbf{r} \times \mathbf{D})(\mathbf{n} \cdot \mathbf{E}) - (\mathbf{r} \times \mathbf{n})(\mathbf{E} \cdot \mathbf{D}) \right) ds$$

where **r** is a radius vector of the point of integration.

The torque vector is parallel to *z*-axis in planar case, and is identically equal to zero in axisymmetric one. The torque is considered relative to the origin of the coordinate system. The torque relative to any other arbitrary point can be obtained by adding extra term of $\mathbf{F} \times \mathbf{r}_0$, where **F** is the total force and $\mathbf{r}_0$ is the radius vector of the point.

- Electric field energy

$$W = \frac{1}{2}\int (\mathbf{E} \cdot \mathbf{D})dV .$$

For magnetostatic problems:

- Total magnetostatic force acting on bodies contained in a particular volume

$$\mathbf{F} = \frac{1}{2}\oint \left(\mathbf{H}(\mathbf{n} \cdot \mathbf{B}) + \mathbf{B}(\mathbf{n} \cdot \mathbf{H}) - \mathbf{n}(\mathbf{H} \cdot \mathbf{B})\right)ds ,$$

where integral is evaluated over the boundary of the volume, and **n** denotes the vector of the outward unit normal.

- Total torque of magnetic forces acting on bodies contained in a particular volume

$$\mathbf{T} = \frac{1}{2}\oint \left((\mathbf{r} \times \mathbf{H})(\mathbf{n} \cdot \mathbf{B}) + (\mathbf{r} \times \mathbf{B})(\mathbf{n} \cdot \mathbf{H}) - (\mathbf{r} \times \mathbf{n})(\mathbf{H} \cdot \mathbf{B})\right)ds ,$$

where **r** is a radius vector of the point of integration.

The torque vector is parallel to *z*-axis in the planar case, and is identically equal to zero in the axisymmetric one. The torque is considered relative to the origin of the coordinate system. The torque relative to any other arbitrary point can be obtained by adding extra term of $\mathbf{F} \times \mathbf{r}_0$, where F is the total force and $\mathbf{r}_0$ is the radius vector of the point.

- Magnetic field energy

$$W = \frac{1}{2}\int (\mathbf{H} \cdot \mathbf{B})dV \qquad \text{— linear case;}$$

$$W = \int \left(\int_0^B H(B')dB' \right) dV \qquad \text{— nonlinear case.}$$

- Flux linkage per one turn of the coil

$$\Psi = \frac{\oint A ds}{S}$$ — for planar case;

$$\Psi = \frac{2\pi \oint rA ds}{S}$$ — for axisymmetric case;

the integral has to be evaluated over a cross section of the coil, and S is the area of the cross section.

For time-harmonic electromagnetic problems:

- Complex magnitude of electric current through a particular surface

  $$I = \int j ds$$

  and also its source and eddy components I_0 and I_e.
- Time average and peak Joule heat in a volume

  $$Q = \int g^{-1} j^2 dV \,.$$
- Time average and peak magnetic field energy

  $$W = \frac{1}{2}\int (\mathbf{H} \cdot \mathbf{B}) dV \,.$$
- Time average and peak power flow through the given surface (Poynting vector flow)

  $$S = \int (\mathbf{S} \cdot \mathbf{n}) ds \,.$$
- Time average and oscillating part of Maxwell force acting on bodies contained in a particular volume

  $$\mathbf{F} = \frac{1}{2}\oint \big(\mathbf{H}(\mathbf{n} \cdot \mathbf{B}) + \mathbf{B}(\mathbf{n} \cdot \mathbf{H}) - \mathbf{n}(\mathbf{H} \cdot \mathbf{B})\big) ds \,,$$

  where integral is evaluated over the boundary of the volume, and **n** denotes the vector of the outward unit normal.
- Time average and peak Maxwell force torque acting on bodies contained in a particular volume

  $$\mathbf{T} = \frac{1}{2}\oint \big((\mathbf{r} \times \mathbf{H})(\mathbf{n} \cdot \mathbf{B}) + (\mathbf{r} \times \mathbf{B})(\mathbf{n} \cdot \mathbf{H}) - (\mathbf{r} \times \mathbf{n})(\mathbf{H} \cdot \mathbf{B})\big) ds \,,$$

where **r** is a radius vector of the point of integration.

- Time average and oscillating part of Lorentz force acting on conductors contained in a particular volume

  $$\mathbf{F} = \int \mathbf{j} \times \mathbf{B} \, dV \,.$$

- Time average and peak Lorentz force torque acting on bodies contained in a particular volume

  $$\mathbf{T} = \int \mathbf{r} \times \left(\mathbf{j} \times \mathbf{B} \right) dV \,,$$

  where **r** is a radius vector of the point of integration.

  The torque vector is parallel to *z*-axis in the planar case, and is identically equal to zero in the axisymmetric one. The torque is considered relative to the origin of the coordinate system. The torque relative to any other arbitrary point can be obtained by adding extra term of $\mathbf{F} \times \mathbf{r}_0$, where **F** is the total force and $\mathbf{r}_0$ is the radius vector of the point.

Note. The Maxwell force incorporates both the force acting on ferromagnetic bodies and Lorentz force, which acts only on conductors. If the first component is negligible or is not considered, we recommend calculating the electromagnetic force as Lorentz force. Its precision is less sensitive to the contour path, and you can simply select conductors via block selection to calculate the force. With Maxwell force, this method leads to very rough results, and you are recommended to avoid coinciding of your contour parts and material boundaries, as described earlier in this chapter.

For problems of current flow:

- Electric current through a given surface

  $$I = \int \mathbf{j} \cdot \mathbf{n} \, ds \,,$$

  where **n** denotes the vector of the unit normal.

- Power losses in a volume

  $$W = \int \mathbf{E} \cdot \mathbf{j} \, dV \,.$$

For heat transfer problems:

- Heat flux through an arbitrary closed or unclosed surface

  $$\Phi = \int \mathbf{F} \cdot \mathbf{n} ds ,$$

  where **n** denotes the unit vector of normal to the surface.

No integral quantities are available for stress analysis.

Data Tables

QuickField can display the field data at discrete points, distributed along the currently selected contour, in table view. To open new table window, choose **Table** in the **View** menu or context (right mouse button) menu in field picture window, in which the contour is already defined.

In table view, you can:

- Select the list of shown quantities (table columns). Choose **Columns** in **View** or context menu.
- Select how the points are distributed along the contour (table rows). Choose **Rows** in **View** or context menu.
- Insert additional rows at specified distance from the beginning of the contour. Choose **Insert** in **Edit** or context menu.
- Copy the set of rows or the whole table to Windows clipboard. In latter case (when all of the rows are selected), column headers are also copied. To copy the header only, click the right mouse button within the header and choose **Copy Header** from the context menu.

Plots and Tables versus Time

Time Plot

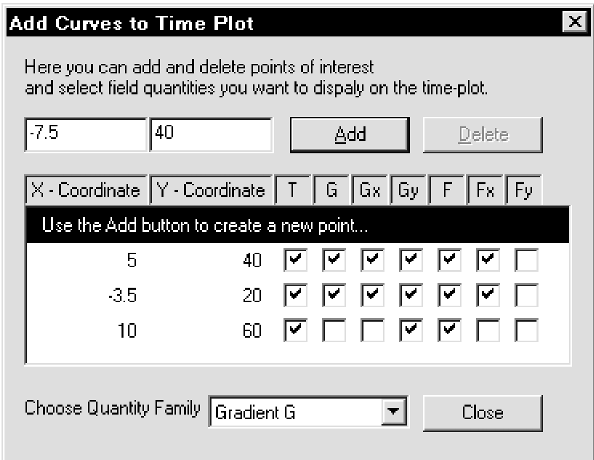
With a transient problem, the QuickField postprocessor can plot dependencies of various field quantities vs. time. To open a new time plot window, choose the **Time Plot** command

- . If you choose the **Time Plot** command in the View menu of field picture window, an empty time plot window opens and the **Add Curves to Time Plot** dialog appears.
- Alternatively right click in the field picture window - a curve for the point you clicked will automatically be added to the time plot. You can obtain the same result by choosing **Time Plot** command in calculator window's context menu, where local values are displayed.

Time plot window can display curves for several points. In its turn, each point can have several curves for different field quantities as long as these quantities belong to the same family: temperature, temperature gradient, or heat flux. You use the time plot control dialog to manipulate points and curves.

Time Plot Control

The **Time Plot Curves** command in the **View** menu or context menu in time plot brings the time plot curves dialog up.

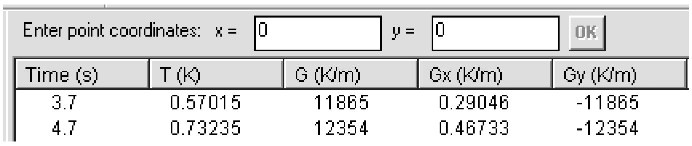


To add a new point, click the very first row in the table, then type coordinates in the boxes above and click the **Add** button. When choosing a point in the list you can change its coordinates and switch on and off associated curves.

The drop-down list at the bottom allows you switch between the curve families. Alternatively you can choose the family in context (right mouse click) menu in time plot window.

Time Dependencies Table

All the field quantities for a given point versus time can be displayed in a table. To open the table choose the **Time Table** command in **View** menu or context menu.


Enter point coordinates: x = 0 y = 0 OK

Time (s)	T (K)	G (K/m)	Gx (K/m)	Gy (K/m)
3.7	0.57015	11865	0.29046	-11865
4.7	0.73235	12354	0.46733	-12354

Initially, the origin becomes a tabulated point. To choose another point, type Cartesian coordinates in the boxes above table and click the **OK** button.

Controlling Legend Display

The legend for the color map shows the correspondence between colors and number; and for X-Y plot—between curves and quantities.

To switch the legend display on or off, click **Legend** in the **View** or context menu in field picture or X-Y plot window.

Printing the Postprocessor Pictures

You can directly print the field picture or X-Y plot to your local or network printer, just as you see the model in the window, with the same zooming and discretization visibility.

- To print the picture, click **Print** in the **File** menu. You will have an option to choose the printer and set up the picture, such as paper size and orientation, before printing will occur.
- To preview the output before printing, click **Print Preview** in the **File** menu. To see how the picture will appear on a printer of your choice, click **Print Setup** before.

Copying the Postprocessor Pictures

You can copy the field picture or X-Y plot, as you see it in the window, to clipboard, for subsequent including it to your paper or report in any word-processing or desktop publishing utility.

- To copy the picture, click **Copy Picture** in the **Edit** menu, or press CTRL+INS.
- Switch to the application where you want to paste the picture and click **Paste** in the **Edit** menu, or press SHIFT+INS.

CHAPTER 8

Theoretical Description

The objective of this chapter is to outline the theories on which the QuickField finite element analysis system is based. The chapter contains underlying mathematical equations, and considers various physical conditions and the ways how they are implemented in QuickField.

QuickField solves 2D boundary value problems for elliptic partial differential equation for either scalar or one-component vector potential. It also solves 2D solid stress analysis problems (plane stress, plane strain, axisymmetric stress). There are three main classes of 2D problems: plane, plane-parallel and axisymmetric. Plane problems usually arise when describing heat transfer processes in thin plates. They are solved in planar rectangular coordinate system. Plane-parallel problems use right-handed Cartesian coordinate system *xyz*. It is assumed that neither geometric shape and properties of material nor field sources vary in *z*-direction. The problem is described, solved and the results are analyzed in *xy*-plane, which we will call the *plane of model*. Axisymmetric problems are formulated in cylindrical coordinate system *zr*θ. The order of axes is chosen for conformity with the plane-parallel case. Physical properties and field sources are assumed to not depend on the angle coordinate. All operations with the model are done in *zr*-plane (more precise in a half plane $r \geq 0$). *Z*-axis is assumed to be horizontal and directed to the right, *r*-axis is directed up.

The geometric configuration of the problem is defined as a set of curved polygonal subregions in the plane of model. Each region corresponds to a domain with a particular set of physical properties. We will use term *blocks* for polygonal subregions, term *edges* for line segments and circular arcs that constitute their boundaries and term *vertices* for ends of edges and for isolated points. Those edges that separate whole problem region from other part of the plane, where no field is calculated, constitutes the *outward* boundary of the region. Other edges constitute *inner* boundaries.

Below you can find detailed mathematical formulations for magnetic, electrostatic, current flow, heat transfer, and stress analysis problems.

Magnetostatics

QuickField can solve both linear and nonlinear magnetic problems. Magnetic field may be induced by the concentrated or distributed currents, permanent magnets or external magnetic fields.

The magnetic problem is formulated as the Poisson's equation for vector magnetic potential **A** (**B = curl A**, **B**—magnetic flux density vector). The flux density is assumed to lie in the plane of model (*xy* or *zr*), while the vector of electric current density **j** and the vector potential **A** are orthogonal to it. Only j_z and A_z in planar or j_θ and A_θ in axisymmetric case are not equal to zero. We will denote them simply j and A. The equation for planar case is

$$\frac{\partial}{\partial x}\left(\frac{1}{\mu_y}\frac{\partial A}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{1}{\mu_x}\frac{\partial A}{\partial y}\right)=-j+\left(\frac{\partial H_{cy}}{\partial x}-\frac{\partial H_{cx}}{\partial y}\right);$$

and for axisymmetric case is

$$\frac{\partial}{\partial r}\left(\frac{1}{r\mu_z}\frac{\partial (rA)}{\partial r}\right)+\frac{\partial}{\partial z}\left(\frac{1}{\mu_r}\frac{\partial A}{\partial z}\right)=-j+\left(\frac{\partial H_{cr}}{\partial z}-\frac{\partial H_{cz}}{\partial r}\right).$$

where components of magnetic permeability tensor μ_x and μ_y (μ_z and μ_r), components of coercive force vector H_{cx} and H_{cy} (H_{cz} and H_{cr}), and current density j are constants within each block of the model.

Note. Isotropic ($\mu_x = \mu_y$ or $\mu_z = \mu_r$) but field dependent permeability is assumed in nonlinear case. Magnetization characteristic of material is described by the *B-H* curve.

Field Sources

The field sources can be specified in blocks, at the edges or at the individual vertices of the model. Possible field sources include space, surface and linear electric currents and permanent magnets. The coercive force is chosen to be primary characteristic for the permanent magnets.

A point source in the *xy*-plane describes a linear current in out-of-plane direction. In axisymmetric case the point source represents the current in a thin ring around the axis of symmetry. Edge-bound source in the plane of model represents a surface current in three-dimensional world. It is specified by the Neumann boundary condition for the edge. The space current is described either by the electric current density or total number of ampere-turns associated with the block density associated with the block. Current density in a coil can be obtained from the equation

$$j = \frac{n \cdot I}{S},$$

where n is a number of turns, I is a total current, and S is a cross-sectional area of the coil.

Several blocks with the same number of ampere-turns specified can be considered as connected in series. In that case current density in each block would be calculated as common total ampere-turns divided by the square of the block.

In axisymmetric case if total number of ampere-turns is specified resulting current density could be described as varies as 1/r, where r is a radius coordinate of the point. This approach allows simulate massive spiral coils.

Boundary Conditions

The following boundary conditions can be specified at outward and inner boundaries of the region.

Dirichlet condition specifies a known value of vector magnetic potential A_0 at the vertex or at the edge of the model. This boundary condition defines normal component of the flux density vector. It is often used to specify vanishing value of this component, for example at the axis of symmetry or at the distant boundary. QuickField also supports the Dirichlet condition with a function of coordinates, it has the form

$$A_0 = a + bx + cy \qquad \text{— for planar problems;}$$

$$rA_0 = a + bzr + \frac{cr^2}{2} \qquad \text{— for axisymmetric problems.}$$

Parameters a, b and c are constants for each edge, but can vary from one piece of the boundary to another. This approach allows you to model an uniform external field by specifying non zero normal component of the flux density at arbitrary straight boundary segment.

Let α be an elevation angle of the segment relative to the horizontal axis (x in planar or z in axisymmetric case). Then in both plane and axisymmetric cases the normal flux density is

$$B_n = c\sin\alpha + b\cos\alpha \ .$$

Here we assume right-hand direction of positive normal vector.

Choice of constant terms a for different edges has to satisfy the continuity conditions for function A_0 at all edges' junction points.

Note. For problem to be defined correctly the Dirichlet condition has to be specified at least at one point. If the region consists of two or more disjoint subregions, the Dirichlet conditions have to be specified at least at one point of the each part. Zero Dirichlet condition is defaulted at the axis of rotation for the axisymmetric problems.

Neumann condition has the following form

$$H_t = \sigma \qquad \text{— at outward boundaries,}$$

$$H_t^+ - H_t^- = \sigma \qquad \text{— at inner boundaries,}$$

where H_t is a tangent component of magnetic field intensity, "+" and "–" superscripts denote quantities to the left and to the right side of the boundary and σ is a linear density of the surface current. If σ value is zero, the boundary condition is called homogeneous. This kind of boundary condition is often used at an outward boundary of the region that is formed by the plane of magnetic antisymmetry of the problem (opposite sources in symmetrical geometry). The homogeneous Neumann condition is the natural one, it is assumed by default at all outward boundary parts where no explicit boundary condition is specified.

Note. Zero Dirichlet condition is defaulted at the axis of rotation for the axisymmetric problems.

If the surface electric current is to be specified at the plane of problem symmetry and this plane forms the outward boundary of the region, the current density has to be halved.

Zero flux boundary condition is used to describe superconducting materials that are not penetrated by the magnetic field. Vector magnetic potential is a constant within such superconducting body ($rA = const$ in axisymmetric case), therefore superconductor's interior can be excluded from the consideration and the constant potential condition can be associated with its surface.

Note. If the surface of a superconductor has common points with any Dirichlet edge, the whole surface has to be described by the Dirichlet condition with an appropriate potential value.

Permanent Magnets

Since the coercive force is considered in QuickField to be the piecewise constant function, its contribution to the equation is equivalent to surface currents which flow along the surface of the permanent magnet in direction orthogonal to the model plane. The density of such effective current is equal to jump of the tangent component of the coercive force across the magnet boundary. For example, rectangular magnet with the coercive force $\mathbf{H}_c$ directed along x-axis can be replaced by two oppositely directed currents at its upper and lower surfaces. The current density at the upper edge is numerically equal to H_c, and $-H_c$ at the lower edge.

Therefore, the permanent magnet can be specified by either coercive force or Neumann boundary conditions at its edges. You can choose more convenient and obvious way in each particular case.

Permanent magnets with nonlinear magnetic properties need some special consideration. Magnetic permeability is assumed to be defined by the following equation

$$B = \mu(B)(H + H_c); \qquad \mu(B) = \frac{B}{H + H_c}.$$

It must be pointed out that $\mu(B)$ dependence is different from the analogous curve for the same material but without permanent magnetism. If the real characteristic for the magnet is not available for you, it is possible to use row material curve as an approximation. If you use such approximation and magnetic field value inside magnet is much smaller than its coercive force, it is recommended to replace the coercive force by the following effective value

$$H_c' = \frac{1}{\mu(B_r)} B_r ,$$

where B_r is remanent induction.

Calculated Physical Quantities

For magnetostatic problems the QuickField postprocessor calculates the following set of local and integral physical quantities.

Local quantities:

- Vector magnetic potential A (flux function rA in axisymmetric case);
- Vector of the magnetic flux density **B = curl A**

  $$B_x = \frac{\partial A}{\partial y}, \qquad B_y = -\frac{\partial A}{\partial x} \qquad \text{— for planar case;}$$

  $$B_z = \frac{1}{r}\frac{\partial (rA)}{\partial r}, \qquad B_r = -\frac{\partial A}{\partial z} \qquad \text{— for axisymmetric case;}$$

- Vector of magnetic field intensity $\mathbf{H} = \mu^{-1}\mathbf{B}$, where μ is the magnetic permeability tensor.

Integral quantities:

- Total magnetostatic force acting on bodies contained in a particular volume

  $$\mathbf{F} = \frac{1}{2}\oint \left(\mathbf{H}(\mathbf{n}\cdot\mathbf{B}) + \mathbf{B}(\mathbf{n}\cdot\mathbf{H}) - \mathbf{n}(\mathbf{H}\cdot\mathbf{B})\right) ds ,$$

  where integral is evaluated over the boundary of the volume, and **n** denotes the vector of the outward unit normal.

- Total torque of magnetic forces acting on bodies contained in a particular volume

$$\mathbf{T} = \frac{1}{2}\oint\left((\mathbf{r}\times\mathbf{H})(\mathbf{n}\cdot\mathbf{B}) + (\mathbf{r}\times\mathbf{B})(\mathbf{n}\cdot\mathbf{H}) - (\mathbf{r}\times\mathbf{n})(\mathbf{H}\cdot\mathbf{B})\right)ds\ ,$$

where **r** is a radius vector of the point of integration.

The torque vector is parallel to *z*-axis in the planar case, and is identically equal to zero in the axisymmetric one. The torque is considered relative to the origin of the coordinate system. The torque relative to any other arbitrary point can be obtained by adding extra term of $\mathbf{F}\times\mathbf{r}_0$, where **F** is the total force and $\mathbf{r}_0$ is the radius vector of the point.

- Magnetic field energy

$$W = \frac{1}{2}\int(\mathbf{H}\cdot\mathbf{B})dV$$ — linear case;

$$W = \int\left(\int_0^B H(B')dB'\right)dV$$ — nonlinear case.

- Flux linkage per one turn of the coil

$$\Psi = \frac{\oint Ads}{S}$$ — for planar case;

$$\Psi = \frac{2\pi\oint rAds}{S}$$ — for axisymmetric case;

the integral has to be evaluated over the cross section of the coil, and *S* is the area of the cross section.

For planar problems all integral quantities are considered per unit length in *z*-direction.

The domain of integration is specified in the plane of the model as a closed contour consisting of line segments and circular arcs.

Inductance Calculation

To get self inductance of a coil, leave the current on in this coil only and make sure that all other currents are turned off. After solving the problem go to the Postprocessor and obtain flux linkage for the contour coinciding with the cross section of the coil. Once you've done that, the inductance of the coil can be obtained from the following equation:

$$L = \frac{n\Psi}{I},$$

where n is a number of turns in the coil, Ψ is a flux linkage, j is a current per one turn of the coil.

Mutual inductance between two coils can be obtained in a similar way. The only difference from the previous case is that electric current has to be turned on in one coil, and the flux linkage has to be evaluated over the cross section of another.

$$L_{12} = \frac{n_2 \Psi_2}{I_1}$$

In plane-parallel case every coil has to be represented by at least two conductors with equal but opposite currents. In some cases both conductors are modeled, in other cases only one of two conductors is included in the model and the rest is replaced by the boundary condition $A = 0$ at the plane of symmetry. If the magnetic system is symmetric, the inductance can be obtained based on the flux linkage for one of the conductors only. The result has to be then multiplied by a factor of two to account for the second conductor. If the model is not symmetric, then the total inductance can be obtained by adding up the analogous terms for each conductor. Note that the current should be turned on in all conductors corresponding to one coil.

In plane-parallel case the inductance is calculated per unit length in z-direction.

Time-Harmonic Magnetic Field

Time-harmonic electromagnetic analysis is the study of electric and magnetic fields arising from the application of an alternating (AC) current source, or an imposed alternating external field.

Variation of the field with respect to time is assumed to be sinusoidal. All field components and electric currents vary with time like

$$z = z_0 \cos(\omega t + \phi_z),$$

where z_0 is a peak value of z, ϕ_z — its phase angle, and ω — the angular frequency.

Complex representation of harmonic time dependency facilitates multiple phase analysis based on one complex solution. Real and imaginary parts of a complex quantity

$$z = z_0 e^{i(\omega t + \phi_z)},$$

have phase angles shifted by 90 degrees, and their linear combination may be used to represent any arbitrary phase angle.

Depending on the phase shift between two oscillating components of a vector, the vector can rotate clockwise or counterclockwise, or oscillate along certain direction. Generally, the end of such a vector draws an ellipse. The semimajor axis of the ellipse corresponds to the peak value of the vector. The ratio between minor and major axes of the ellipse defines the coefficient of polarization. The coefficient of polarization is assumed to be positive for the counterclockwise and negative for the clockwise rotation. Zero coefficient corresponds to the linear polarization.

Total current in a conductor can be considered as a combination of a source current produced by the external voltage and an eddy current induced by the oscillating magnetic field

$$\mathbf{j} = \mathbf{j}_0 + \mathbf{j}_{eddy}$$

The problem is formulated as a partial differential equation for the complex amplitude of vector magnetic potential **A** (**B = curl A**, **B**—magnetic flux density vector). The flux density is assumed to lie in the plane of model (*xy* or *zr*), while the vector of electric current density **j** and the vector potential **A** are orthogonal to it. Only j_z and A_z in planar or j_θ and A_θ in axisymmetric case are not equal to zero. We will denote them simply *j* and *A*. The equation for planar case is

$$\frac{\partial}{\partial x}\left(\frac{1}{\mu_y}\frac{\partial A}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{1}{\mu_x}\frac{\partial A}{\partial y}\right) - i\,\omega g A = -j_0;$$

and for axisymmetric case is

$$\frac{\partial}{\partial r}\left(\frac{1}{r\mu_z}\frac{\partial (rA)}{\partial r}\right) + \frac{\partial}{\partial z}\left(\frac{1}{\mu_r}\frac{\partial A}{\partial z}\right) - i\,\omega g A = -j_0\,.$$

where electric conductivity *g* and components of magnetic permeability tensor μ_x and μ_y (μ_z and μ_r) are constants within each block of the model. Source current density j_0

is assumed to be constant within each model block in planar case and vary as $1/r$ in axisymmetric case.

The described formulation ignores displacement current density term $\partial \mathbf{D}/\partial t$ in the Ampere's Law. Typically the displacement current density is not significant until the operating frequency approaches the MHz range.

Note. Permanent magnets and nonlinear magnetic properties cannot be simulated in a time-harmonic analysis. Since the entire field must vary sinusoidally, this would prevent permanent magnets or effects of saturation from being simulated using the harmonic analysis. Permanent magnets supply a constant flux to the system. Saturation of the material indicates that as the field intensity increases, the flux density in the material would not be able to follow the same sinusoidal behavior.

Field Sources

The field sources can be specified in the blocks, at the edges or at the individual vertices of the model. Possible field sources include space, surface and linear electric currents and voltages applied to conductive areas.

A point source in the *xy*-plane corresponds to a linear current in out-of-plane direction. In axisymmetric case the point source represents the current in a thin ring around the axis of symmetry. Edge-bound source in the plane of model represents a surface current in three-dimensional world. It is specified by the Neumann boundary condition for the edge.

There are several ways to specify space-distributed electric current. In a massive conductor, you can specify either a total current or a voltage applied to the conductor. In planar problems, voltage drop is specified per unit depth of the model, and in axisymmetric case voltage is assumed per one turn around the axis of symmetry. Nonzero voltage applied to a conductor in axisymmetric problem means that the conductor has a radial cut, and the voltage is applied to sides of the cut. In practice this option could be used to describe known voltage applied to massive spiral wiring, in which case the total voltage drop for the coil should be divided by number of turns in the coil.

Several blocks with the same value of total current or voltage applied can be considered as connected in series. In that case each conductor carries the same total current, and voltage (if any) is applied to the terminals of the whole group of conductors connected in series.

Note. The meanings of zero total current and zero voltage applied to a conductor are very different. Zero voltage means that the conductor's ends are short circuit, and zero value of the total current means open ends of the conductor.

Field source could also be specified in non-conductive areas. This option is useful to specify current in coils made of thin wire, where skin effect is insignificant. You can specify either a total current or a current density, whichever is easier to calculate in a specific case. Current density in a coil can be obtained from the equation

$$j = \frac{n \cdot I}{S},$$

where n is a number of turns, I is a total current, and S is a cross-sectional area of the coil.

Note. In order to properly model thin wire coils, the source current density j_0 in non-conductive areas is assumed to be uniform in both plane and axisymmetric cases. Its behavior is different for massive conductors, where source current density varies as $1/r$ in axisymmetric case.

Boundary Conditions

The following boundary conditions can be specified at outward and inner boundaries of the region.

Dirichlet condition specifies a known value of vector magnetic potential A_0 at the vertex or at the edge of the model. This boundary condition defines normal component of the flux density vector. It is often used to specify vanishing value of this component, for example at the axis of symmetry or at the distant boundary. QuickField also supports the Dirichlet condition with a function of coordinates, it has the form

$$A_0 = a + bx + cy$$ — for planar problems;

$$rA_0 = a + bzr + \frac{cr^2}{2}$$ — for axisymmetric problems.

Parameters a, b and c are constants for each edge, but can vary from one piece of the boundary to another. This approach allows you to model an uniform external field by specifying non zero normal component of the flux density at arbitrary straight boundary segment.

Let α be an elevation angle of the segment relative to the horizontal axis (x in planar or z in axisymmetric case). Then in both plane and axisymmetric cases the normal flux density is

$$B_n = c\sin\alpha + b\cos\alpha \ .$$

Here we assume right-hand direction of positive normal vector.

Choice of constant terms a for different edges has to satisfy the continuity conditions for function A_0 at all edges' junction points.

Neumann condition has the following form

$$H_t = \sigma \qquad \text{— at outward boundaries,}$$

$$H_t^+ - H_t^- = \sigma \qquad \text{— at inner boundaries,}$$

where H_t is a tangent component of magnetic field intensity, "+" and "−" superscripts denote quantities to the left and to the right side of the boundary and σ is a linear density of the surface current. If σ value is zero, the boundary condition is called homogeneous. This kind of boundary condition is often used at an outward boundary of the region that is formed by the plane of magnetic antisymmetry of the problem (opposite sources in symmetrical geometry). The homogeneous Neumann condition is the natural one, it is assumed by default at all outward boundary parts where no explicit boundary condition is specified.

Note. Zero Dirichlet condition is defaulted at the axis of rotation for the axisymmetric problems.

If the surface electric current is to be specified at the plane of problem symmetry and this plane forms the outward boundary of the region, the current density has to be halved.

Zero flux boundary condition is used to describe superconducting materials that are not penetrated by the magnetic field. Vector magnetic potential is a constant within such superconducting body ($rA = const$ in axisymmetric case), therefore superconductor's interior can be excluded from the consideration and the constant potential condition can be associated with its surface.

Note. If the surface of a superconductor has common points with any Dirichlet edge, the whole surface has to be described by the Dirichlet condition with an appropriate potential value.

Calculated Physical Quantities

The following local and integral physical quantities are calculated in the process of harmonic magnetic field analysis.

Local quantities:

- Complex amplitude of vector magnetic potential A (flux function rA in axisymmetric case);
- Complex amplitude of voltage U applied to the conductor;
- Complex amplitude of total current density $j = j_0 + j_{\text{eddy}}$, source current density j_0 and eddy current density $j_{\text{eddy}} = -i\omega gA$;
- Complex vector of the magnetic flux density **B = curl A**

  $$B_x = \frac{\partial A}{\partial y}, \qquad B_y = -\frac{\partial A}{\partial x} \qquad \text{— for planar case;}$$

  $$B_z = \frac{1}{r}\frac{\partial(rA)}{\partial r}, \qquad B_r = -\frac{\partial A}{\partial z} \qquad \text{— for axisymmetric case;}$$

- Complex vector of magnetic field intensity $\mathbf{H} = \mu^{-1}\mathbf{B}$, where μ is the magnetic permeability tensor;
- Time average and peak Joule heat density $Q = g^{-1} j^2$;
- Time average and peak magnetic field energy density $w = (\mathbf{B} \cdot \mathbf{H})/2$;
- Time average Poynting vector (local power flow) $\mathbf{S} = \mathbf{E} \times \mathbf{H}$;
- Time average Lorentz force density vector $\mathbf{F} = \mathbf{j} \times \mathbf{B}$;
- Magnetic permeability μ (its largest component in anisotropic media);
- Electric conductivity g.

Integral quantities:

- Complex magnitude of electric current through a particular surface

  $$I = \int jds$$

  and its source and eddy components I_0 and I_e.
- Time average and peak Joule heat in a volume

$$Q = \int g^{-1} j^2 dV \ .$$

- Time average and peak magnetic field energy

$$W = \frac{1}{2}\int (\mathbf{H} \cdot \mathbf{B}) dV \ .$$

- Time average and peak power flow through the given surface (Poynting vector flow)

$$S = \int (\mathbf{S} \cdot \mathbf{n}) ds \ .$$

- Time average and oscillating part of Maxwell force acting on bodies contained in a particular volume

$$\mathbf{F} = \frac{1}{2}\oint \big(\mathbf{H}(\mathbf{n} \cdot \mathbf{B}) + \mathbf{B}(\mathbf{n} \cdot \mathbf{H}) - \mathbf{n}(\mathbf{H} \cdot \mathbf{B})\big) ds \ ,$$

where integral is evaluated over the boundary of the volume, and **n** denotes the vector of the outward unit normal.

- Time average and peak Maxwell force torque acting on bodies contained in a particular volume

$$\mathbf{T} = \frac{1}{2}\oint \big((\mathbf{r} \times \mathbf{H})(\mathbf{n} \cdot \mathbf{B}) + (\mathbf{r} \times \mathbf{B})(\mathbf{n} \cdot \mathbf{H}) - (\mathbf{r} \times \mathbf{n})(\mathbf{H} \cdot \mathbf{B})\big) ds \ ,$$

where **r** is a radius vector of the point of integration.

- Time average and oscillating part of Lorentz force acting on conductors contained in a particular volume

$$\mathbf{F} = \int \mathbf{j} \times \mathbf{B} \, dV \ .$$

- Time average and peak Lorentz force torque acting on bodies contained in a particular volume

$$\mathbf{T} = \int \mathbf{r} \times \big(\mathbf{j} \times \mathbf{B}\big) dV \ ,$$

where **r** is a radius vector of the point of integration.

The torque vector is parallel to *z*-axis in the planar case, and is identically equal to zero in the axisymmetric one. The torque is considered relative to the origin of the coordinate system. The torque relative to any other arbitrary point can be obtained by adding extra term of $\mathbf{F} \times \mathbf{r}_0$, where **F** is the total force and $\mathbf{r}_0$ is the radius vector of the point.

Note. Magnetic field produces forces acting on the current carrying conductors and on the ferromagnetic bodies. The force acting on conductors is known as Lorentz force, while the Maxwell force incorporates both components.

The domain of integration is specified in the plane of the model as a closed contour consisting of line segments and circular arcs.

Impedance Calculation

Impedance in time-harmonic electromagnetic analysis is a complex coefficient between complex values of current and voltage, $V = ZI$. Its real part represents active resistance of the conductor, calculated with the skin effect taken into account. The imaginary part of the impedance is the inductance multiplied by the angular frequency ω.

$$Z = R + i\omega L\,.$$

As values of voltage and current in any conductor are easily accessible in the postprocessor, you can determine the impedance by dividing voltage by current using complex arithmetic. Let V and I be peak values of voltage and current, and ϕ_V and ϕ_I be phases of those quantities. Then the active resistance could be calculated as

$$R = \frac{V}{I}\cos\left(\phi_V - \phi_I\right),$$

and the inductance as

$$L = \frac{V}{I \cdot 2\pi f}\sin\left(\phi_V - \phi_I\right).$$

To get mutual inductance between two conductors, you can specify nonzero total current in one of them, make the ends of the other open (applying zero total current), and measure the voltage induced in the second conductor by the current in the first one.

Note. As in planar case voltage is applied and measured per unit length, the impedance is also calculated per unit length in *z*-direction.

Electrostatics

Electrostatic problems are described by the Poisson's equation for scalar electric potential U (**E = –grad***U*, **E**—electric field intensity vector). The equation for planar case is

$$\frac{\partial}{\partial x}\left(\varepsilon_x \frac{\partial U}{\partial x}\right)+\frac{\partial}{\partial y}\left(\varepsilon_y \frac{\partial U}{\partial y}\right)=-\rho\,,$$

and for axisymmetric case is

$$\frac{1}{r}\frac{\partial}{\partial r}\left(\varepsilon_r r \frac{\partial U}{\partial r}\right)+\frac{\partial}{\partial z}\left(\varepsilon_z \frac{\partial U}{\partial z}\right)=-\rho\,,$$

where components of electric permittivity tensor ε_x, ε_y or ε_z, ε_r and electric charge density ρ are constants within each block of the model.

Field Sources

QuickField provides possibility to specify electric charges located in the blocks, at the edges or at the individual vertices of the model. The electric charge specified at a point of the *xy*-plane corresponds to a charged string which is perpendicular to the plane of the model, and is described by the linear charge density. In axisymmetric case the vertex charge represents a charged circle around the axis of symmetry or a point charge located on the axis. To incorporate both these cases a total charge value is associated with the vertex. For the charged circle the total charge is connected with its linear density by the relationship $q = 2\pi r\cdot\rho$. Edge-bound charge in the plane of model represents a surface-bound charge in three-dimensional world. It is described by surface charge density and is specified by the Neumann boundary condition for the edge. The charge density associated with a block is equivalent to the space charge.

Boundary Conditions

The following boundary conditions can be specified at outward and inner boundaries of the region.

Dirichlet condition electric potential: specifies a known value of electric potential U_0 at the vertex or at the edge of the model (for example on a capacitor plate). This kind of boundary condition is also useful at an outward boundary of the region that is formed by the plane of electric antisymmetry of the problem (opposite charges in symmetrical geometry). U_0 value at the edge can be specified as a linear function of coordinates. The function parameters can vary from one edge to another, but have to be adjusted to avoid discontinuities at edges' junction points.

Note. For problem to be defined correctly the Dirichlet condition has to be specified at least at one point. If the region consists of two or more disjoint subregions, the Dirichlet conditions have to be specified at least at one point of every part.

Neumann condition is defined by the following equations:

$D_n = \sigma$ — at outward boundaries,

$D_n^+ - D_n^- = \sigma$ — at inner boundaries,

where D_n is a normal component of electric induction, "+" and "–" superscripts denote quantities to the left and to the right side of the boundary, σ is a surface charge density. If σ value is zero, the boundary condition is called homogeneous. It indicates vanishing of the normal component of electric field intensity vector. This kind of boundary condition is used at an outward boundary of the region that is formed by the symmetry plane of the problem. The homogeneous Neumann condition is the natural one, it is defaulted at all outward boundary parts where no explicit boundary condition is specified.

If the surface-bound charge is to be specified at the plane of problem symmetry and this plane is the outward boundary of the region, the surface charge density has to be halved.

Constant potential boundary condition is used to describe surface of an isolated “floating” conductor that has constant but unknown potential value.

Note. The edge described as possessing constant potential should not have common points with any Dirichlet edge. In that case the constant potential edge has to be described by a Dirichlet condition with appropriate potential value.

Calculated Physical Quantities

For electrostatic problems the QuickField postprocessor calculates the following set of local and integral physical quantities.

Local quantities:

- Scalar electric potential U;
- Vector of electric field intensity $\mathbf{E} = -\mathrm{grad}U$

  $E_x = -\dfrac{\partial U}{\partial x}$, $E_y = -\dfrac{\partial U}{\partial y}$ —for planar case;

  $E_z = -\dfrac{\partial U}{\partial z}$, $E_r = -\dfrac{\partial U}{\partial r}$ —for axisymmetric case;

- Tensor of the gradient of electric field intensity $\mathbf{G} = \mathrm{grad}\mathbf{E}$

  $G_{xx} = \dfrac{\partial E_x}{\partial x}, G_{yy} = \dfrac{\partial E_y}{\partial y}, G_{xy} = \dfrac{1}{2}\left(\dfrac{\partial E_x}{\partial y} + \dfrac{\partial E_y}{\partial x}\right)$ —for planar case;

  $G_{zz} = \dfrac{\partial E_z}{\partial z}, G_{rr} = \dfrac{\partial E_r}{\partial r}, G_{zr} = \dfrac{1}{2}\left(\dfrac{\partial E_z}{\partial r} + \dfrac{\partial E_r}{\partial z}\right)$ —for axisymmetric case;

  and also its principal components G_1 and G_2.
- Vector of electric induction $\mathbf{D} = \varepsilon\mathbf{E}$, where ε is electric permittivity tensor.

Integral quantities:

- Total electric charge in a particular volume

  $$q = \oint \mathbf{D} \cdot \mathbf{n} ds ,$$

  where integral is evaluated over the boundary of the volume, and **n** denotes the vector of the outward unit normal.
- Total electrostatic force acting on bodies contained in a particular volume

  $$\mathbf{F} = \frac{1}{2}\oint (\mathbf{E}(\mathbf{n} \cdot \mathbf{D}) + \mathbf{D}(\mathbf{n} \cdot \mathbf{E}) - \mathbf{n}(\mathbf{E} \cdot \mathbf{D}))ds$$

- Total torque of electrostatic forces acting on bodies contained in a particular volume

  $$\mathbf{T} = \frac{1}{2}\oint ((\mathbf{r}\times\mathbf{E})(\mathbf{n} \cdot \mathbf{D}) + (\mathbf{r}\times\mathbf{D})(\mathbf{n} \cdot \mathbf{E}) - (\mathbf{r}\times\mathbf{n})(\mathbf{E} \cdot \mathbf{D}))ds$$

  where **r** is a radius vector of the point of integration.

The torque vector is parallel to z-axis in planar case, and is identically equal to zero in axisymmetric one. The torque is considered relative to the origin of the coordinate system. The torque relative to any other arbitrary point can be obtained by adding extra term of $\mathbf{F} \times \mathbf{r}_0$, where $\mathbf{F}$ is the total force and $\mathbf{r}_0$ is the radius vector of the point.

- Energy of electric field

$$W = \frac{1}{2}\int (\mathbf{E} \cdot \mathbf{D}) dV \ .$$

For planar problems all integral quantities are considered per unit length in z-direction.

The domain of integration is specified in the plane of the model as a closed contour consisting of line segments and circular arcs.

Capacitance Calculation

There are several ways to calculate capacitance using QuickField. The easiest one of them is based on measuring an electric potential produced by a known charge. To get capacitance of a conductor, put constant potential boundary condition on its surface, specify an arbitrary non zero electric charge in one of the vertices on the surface of the conductor (in fact, the charge will be distributed over the conductor's surface), and turn off all other field sources in the model. Once the problem is solved, go to the Postprocessor and take the value of electric potential somewhere on the surface of the conductor. The capacitance of the conductor can be obtained from the equation

$$C = \frac{q}{U} ,$$

where q is the electric charge and U is the potential of the conductor.

To calculate mutual capacitance between two conductors put a charge on one conductor and measure electric potential on another. Constant potential boundary condition has to be applied to the surfaces of both conductors.

$$C_{12} = \frac{q_1}{U_2} .$$

Other ways of calculating capacitance are demonstrated in example *"Elec1: Microstrip Transmission Line"*.

Current Flow Analysis

QuickField is able to calculate the distribution of electric current in systems of conductors. The problems of current distribution are described by the Poisson's equation for scalar electric potential *U*.

The equation for planar case is

$$\frac{\partial}{\partial x}\left(\frac{1}{\rho_x}\frac{\partial U}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{1}{\rho_y}\frac{\partial U}{\partial y}\right)=0\,,$$

and for axisymmetric case is

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{1}{\rho_r}\frac{\partial U}{\partial r}\right)+\frac{\partial}{\partial z}\left(\frac{1}{\rho_z}\frac{\partial U}{\partial z}\right)=0\,,$$

where components of electric resistivity tensor ρ_x, ρ_y or ρ_z, ρ_r are constant within each model block.

The electric current density **j** can be obtained from the equation $\mathbf{j} = -\rho^{-1}\cdot\mathbf{grad}\ U$, where ρ^{-1} is an inverse tensor of electric resistivity.

Field Sources

With the problems of current flow, the field sources are external currents supplied to the boundary of a conductor. QuickField provides possibility to specify external current density at the edges or at the individual vertices of the model. The current density specified at a point of the *xy*-plane corresponds to a knife-edge current collector which is perpendicular to the plane of the model, and is described by the linear current density. In axisymmetric case the vertex source represents a circular collector around the axis of symmetry or a point collector located on the axis. To incorporate both these cases, a total current value is associated with the vertex. For the circular knife-edge collector the total current value is connected with its linear density by the relationship $I = 2\pi r\cdot\sigma$. Edge-bound current density in the plane of model represents a surface-bound external current density in three-dimensional world. It is specified by the Neumann boundary condition for the edge.

Boundary Conditions

The following boundary conditions can be specified at outward and inner boundaries of the region.

Dirichlet condition electric potential: specifies a known value of electric potential U_0 at the vertex or at the edge of the model. U_0 value at the edge can be specified as a linear function of coordinates. The function parameters can vary from one edge to another, but have to be adjusted to avoid discontinuities at edges' junction points.

Note. For problem to be defined correctly the Dirichlet condition has to be specified at least at one point. If the region consists of two or more disjoint subregions, the Dirichlet conditions have to be specified at least at one point of every part.

Neumann condition is defined by the following equations:

$$j_n = j \qquad \text{— at outward boundaries,}$$

$$j_n^+ - j_n^- = j \qquad \text{— at inner boundaries,}$$

where j_n is a normal component of the current density vector, "+" and "–" superscripts denote quantities to the left and to the right side of the boundary, and j at right hand side is a density of the external current. If j value is zero, the boundary condition is called homogeneous. This kind of boundary condition is used at an outward boundary of the region that is formed by the symmetry plane of the problem. The homogeneous Neumann condition is the natural one, it is defaulted at all outward boundary parts where no explicit boundary condition is specified.

If the surface-bound current density is to be specified at the plane of problem symmetry and this plane is the outward boundary of the region, the surface current density has to be halved.

Constant potential boundary condition is used to describe surface of a conductor having much greater conductivity than the surrounding medium. This conductor is assumed to have constant but unknown potential value.

Note. The edge described as possessing constant potential should not have common points with any Dirichlet edge. In that case the constant potential edge has to be described by the Dirichlet condition with an appropriate potential value.

Calculated Physical Quantities

For problems of current flow, the QuickField postprocessor calculates the following set of local and integral physical quantities.

Local quantities:

- Scalar electric potential U;
- Vector of electric field intensity $\mathbf{E} = -\mathrm{grad}U$

$$E_x = -\frac{\partial U}{\partial x}, \qquad E_y = -\frac{\partial U}{\partial y} \qquad \text{— for planar case;}$$

$$E_z = -\frac{\partial U}{\partial z}, \qquad E_r = -\frac{\partial U}{\partial r} \qquad \text{— for axisymmetric case;}$$

- Vector of current density $\mathbf{j} = \rho^{-1}\mathbf{E}$, where ρ is electric resistivity tensor.

Integral quantities:

- Electric current through a given surface

  $$I = \int \mathbf{j} \cdot \mathbf{n} ds,$$

  where **n** denotes the vector of the unit normal.
- Joule heat produced in a volume

  $$W = \int \mathbf{E} \cdot \mathbf{j} dV.$$

For planar problems all integral quantities are considered per unit length in z direction.

The domain of integration is specified in the plane of the model as a closed contour consisting of line segments and circular arcs.

Heat Transfer

With QuickField you can analyze linear and nonlinear temperature fields in one of two formulations: steady state or transient: heating or cooling of the system.

Heat-transfer equation for linear problems is:

$$\frac{\partial}{\partial x}\left(\lambda_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\lambda_y \frac{\partial T}{\partial y}\right) = -q - c\rho \frac{\partial T}{\partial t} \qquad \text{— planar case;}$$

$$\frac{1}{r}\frac{\partial}{\partial r}\left(\lambda_r r\frac{\partial T}{\partial r}\right)+\frac{\partial}{\partial z}\left(\lambda_z \frac{\partial T}{\partial z}\right)=-q-c\rho\frac{\partial T}{\partial t}$$ — axisymmetric case;

for nonlinear problems:

$$\frac{\partial}{\partial x}\left(\lambda(T)\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\lambda(T)\frac{\partial T}{\partial y}\right)=-q(T)-c(T)\rho\frac{\partial T}{\partial t}$$ — planar case;

$$\frac{1}{r}\frac{\partial}{\partial r}\left(\lambda(T)r\frac{\partial T}{\partial r}\right)+\frac{\partial}{\partial z}\left(\lambda(T)\frac{\partial T}{\partial z}\right)=-q(T)-c(T)\rho\frac{\partial T}{\partial t}$$ — axisymmetric case;

where:

T — temperature;

t — time;

$\lambda_{x(y,z,r)}$ — components of heat conductivity tensor;

$\lambda(T)$ — heat conductivity as a function of temperature approximated by cubic spline (anisotropy is not supported in nonlinear case);

$q(T)$ — volume power of heat sources, in linear case—constant, in nonlinear case—function of temperature approximated by cubic spline.

$c(T)$ — specific heat, in nonlinear case—function of temperature approximated by cubic spline;

ρ — density of the substance.

In steady state case the last term in these equations equals zero.

In linear case all the parameters are constants within each block of the model.

The heat transfer problems for thin plates are very analogous to the plane-parallel problems and we will not discuss them especially.

Heat Sources

QuickField provides possibility to specify the heat sources located in the blocks, at the edges or at the individual vertices of the model. The heat source specified at a point of the *xy*-plane corresponds to a linear string-like heater which is perpendicular to the plane of the model, and is described by the generated power per unit length. In axisymmetric case the vertex heat source represents a heating circle around the axis of symmetry or a point heater located on the axis. To incorporate both these cases a total generated power value is associated with the vertex. For the heating circle the total power is connected with its linear density by the relationship $q = 2\pi r \cdot q_l$. Edge-bound heat source in the plane of model represents a surface heat source in three-dimensional world. It is described by power per unit area and is specified by the Neumann boundary condition for the edge. The volume power density associated with a block corresponds to the volume heat source.

Boundary Conditions

The following boundary conditions can be specified at outward and inner boundaries of the region.

Known temperature boundary condition (known also as boundary condition of the first kind) specifies a known value of temperature T_0 at the vertex or at the edge of the model (for example on a liquid-cooled surface). T_0 value at the edge can be specified as a linear function of coordinates. The function parameters can vary from one edge to another, but have to be adjusted to avoid discontinuities at edges' junction points.

This boundary condition sometimes is called the boundary condition of the first kind.

Heat flux boundary condition (known also as boundary condition of the second kind) is defined by the following equations:

$F_n = -q_s$ — at outward boundaries,

$F_n^+ - F_n^- = -q_s$ — at inner boundaries,

where F_n is a normal component of heat flux density, "+" and "−" superscripts denote quantities to the left and to the right side of the boundary. For inner boundary q_s, denotes the generated power per unit area, for outward boundary it specifies the known value of the heat flux density across the boundary. If q_s, value is zero, the boundary condition is called homogeneous. The homogeneous condition at the outward boundary indicates vanishing of the heat flux across the surface. This type of boundary condition is the natural one, it is defaulted at all outward boundary parts where no explicit boundary condition is specified. This kind of boundary condition is used at an outward boundary of the region which is formed by the symmetry plane of the problem.

If the surface heat source is to be specified at the plane of problem symmetry and this plane constitutes the outward boundary of the region, the surface power has to be halved.

This boundary condition sometimes is called the boundary condition of the second kind.

Convection boundary condition can be specified at outward boundary of the region. It describes convective heat transfer and is defined by the following equation:

$$F_n = \alpha\left(T - T_0\right),$$

where α is a film coefficient, and T_0—temperature of contacting fluid medium. Parameters α and T_0 may differ from part to part of the boundary.

This boundary condition sometimes is called the boundary condition of the third kind.

Radiation boundary condition can be specified at outward boundary of the region. It describes radiative heat transfer and is defined by the following equation:

$$F_n = k_{\mathrm{SB}}\beta\left(T^4 - T_0^4\right),$$

where k_{SB} is a Stephan-Boltsman constant, β is an emissivity coefficient, and T_0—ambient radiation temperature. Parameters β and T_0 may differ from part to part of the boundary.

Note. For heat transfer problem to be defined correctly the known temperature boundary condition, or the convection, or the radiation has to be specified at least at some parts of the boundary.

Constant temperature boundary condition may be used to describe bodies with very high heat conductivity. You can exclude interior of these bodies from the consideration and describe their surface as the constant temperature boundary.

Note. The edge described as possessing constant temperature cannot have common points with any edge where the temperature is specified explicitly. In that case the constant temperature edge has to be described by the boundary condition of the first kind with an appropriate temperature value.

Calculated Physical Quantities

For heat transfer problems the QuickField postprocessor calculates the following set of local and integral physical quantities.

Local quantities:

- Temperature T;
- Vector of the heat flux density $\mathbf{F} = -\lambda\ \mathbf{grad}\ T$

$$F_x = -\lambda_x \frac{\partial T}{\partial x}, \qquad F_y = -\lambda_y \frac{\partial T}{\partial y}$$ — for planar case;

$$F_z = -\lambda_z \frac{\partial T}{\partial z}, \qquad F_r = -\lambda_r \frac{\partial T}{\partial r}$$ — for axisymmetric case;

The postprocessor can calculate the heat flux through an arbitrary closed or unclosed surface

$$\Phi = \int \mathbf{F} \cdot \mathbf{n} ds ,$$

where **n** denotes the unit vector of normal to the surface. The surface is specified by a contour consisting of line segments and circular arcs in the plane of the model.

Stress Analysis

Within QuickField package, the plane stress, the plane strain and the axisymmetric stress models are available with both isotropic and orthotropic materials. The plane stress model is suitable for analyzing structures that are thin in the out-of-plane direction, e.g., thin plates subject to in-plate loading. Out-of-plane direct stress and shear stresses are assumed to be negligible. The plane strain model is formulated by assuming that out-of-plane strains are negligible. This model is suitable for structures that are thick in the out-of-plane direction.

Displacement, Strain and Stress

The displacement field is assumed to be completely defined by the two components of the displacement vector δ in each point:

$$\{\delta\} = \begin{Bmatrix} \delta_x \\ \delta_y \end{Bmatrix}$$ — for plane problems;

$$\{\delta\} = \begin{Bmatrix} \delta_z \\ \delta_r \end{Bmatrix} \qquad \text{— for axisymmetric problems.}$$

Only three components of strain and stress tensors are independent in both plane stress and plane strain cases. The strain-displacement relationship is defined as:

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \dfrac{\partial \delta_x}{\partial x} \\ \dfrac{\partial \delta_y}{\partial y} \\ \dfrac{\partial \delta_x}{\partial y} + \dfrac{\partial \delta_y}{\partial x} \end{Bmatrix}.$$

The corresponding stress components:

$$\{\sigma\} = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}.$$

The axisymmetric problem formulation also includes the out-of-plane direct strain ε_θ, caused by the radial deformation. The strain-displacement relationship is defined as:

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_z \\ \varepsilon_r \\ \varepsilon_\theta \\ \gamma_{rz} \end{Bmatrix} = \begin{Bmatrix} \dfrac{\partial \delta_z}{\partial z} \\ \dfrac{\partial \delta_r}{\partial r} \\ \dfrac{\delta_r}{r} \\ \dfrac{\partial \delta_z}{\partial r} + \dfrac{\partial \delta_r}{\partial z} \end{Bmatrix}.$$

The corresponding stress components:

$$\{\sigma\} = \begin{Bmatrix} \sigma_z \\ \sigma_r \\ \sigma_\theta \\ \tau_{rz} \end{Bmatrix}.$$

The equilibrium equations for the plane problems are:

$$\begin{cases} \dfrac{\partial \sigma_x}{\partial x} + \dfrac{\partial \tau_{xy}}{\partial y} = -f_x \\ \dfrac{\partial \tau_{xy}}{\partial x} + \dfrac{\partial \sigma_y}{\partial y} = -f_y, \end{cases}$$

and for the axisymmetric problems are:

$$\begin{cases} \dfrac{1}{r}\dfrac{\partial (r\sigma_r)}{\partial r} + \dfrac{\partial \tau_{rz}}{\partial z} = -f_r \\ \dfrac{1}{r}\dfrac{\partial (r\tau_{rz})}{\partial r} + \dfrac{\partial \sigma_z}{\partial z} = -f_z, \end{cases}$$

where f_x, f_y and f_z, f_r are components of the volume force vector.

For linear elasticity, the stresses are related to the strains using relationship of the form

$$\{\sigma\} = [D](\{\varepsilon\} - \{\varepsilon_0\}),$$

where $[D]$ is a matrix of elastic constants, and $\{\varepsilon_0\}$ is the initial thermal strain. The specific form of the matrix depends on a particular problem formulation.

For plane stress and isotropic material:

$$[D] = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{1-\nu}{2} \end{bmatrix}.$$

For plane stress and orthotropic material:

$$[D] = \begin{bmatrix} \dfrac{1}{E_x} & -\dfrac{\nu_{yx}}{E_y} & 0 \\ -\dfrac{\nu_{yx}}{E_y} & \dfrac{1}{E_y} & 0 \\ 0 & 0 & \dfrac{1}{G_{xy}} \end{bmatrix}^{-1}.$$

For plane strain and isotropic material:

$$[D] = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \dfrac{\nu}{1-\nu} & 0 \\ \dfrac{\nu}{1-\nu} & 1 & 0 \\ 0 & 0 & \dfrac{1-2\nu}{2(1-\nu)} \end{bmatrix}.$$

For plane strain and orthotropic material:

$$[D] = \begin{bmatrix} \dfrac{1}{E_x} - \dfrac{\nu_{zx}^2}{E_z} & -\dfrac{\nu_{yx}}{E_y} - \dfrac{\nu_{zx}\nu_{zy}}{E_z} & 0 \\ -\dfrac{\nu_{yx}}{E_y} - \dfrac{\nu_{zx}\nu_{zy}}{E_z} & \dfrac{1}{E_y} - \dfrac{\nu_{zy}^2}{E_z} & 0 \\ 0 & 0 & \dfrac{1}{G_{xy}} \end{bmatrix}^{-1}.$$

For axisymmetric problem and isotropic material:

$$[D] = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \dfrac{\nu}{1-\nu} & \dfrac{\nu}{1-\nu} & 0 \\ \dfrac{\nu}{1-\nu} & 1 & \dfrac{\nu}{1-\nu} & 0 \\ \dfrac{\nu}{1-\nu} & \dfrac{\nu}{1-\nu} & 1 & 0 \\ 0 & 0 & 0 & \dfrac{1-2\nu}{2(1-\nu)} \end{bmatrix};$$

For axisymmetric problem and orthotropic material:

$$[D] = \begin{bmatrix} \dfrac{1}{E_z} & -\dfrac{\nu_{rz}}{E_r} & -\dfrac{\nu_{\theta z}}{E_\theta} & 0 \\ -\dfrac{\nu_{rz}}{E_r} & \dfrac{1}{E_r} & -\dfrac{\nu_{\theta r}}{E_\theta} & 0 \\ -\dfrac{\nu_{\theta z}}{E_\theta} & -\dfrac{\nu_{\theta r}}{E_\theta} & \dfrac{1}{E_\theta} & 0 \\ 0 & 0 & 0 & \dfrac{1}{G_{zr}} \end{bmatrix}^{-1}.$$

In all these equations E denotes Young's modulus of the isotropic material; E_x, E_y, E_z, E_r, and E_θ are the Young's moduli of the orthotropic material along the corresponding axes; ν is a Poisson's ratio for isotropic material; ν_{yx}, ν_{zx}, ν_{zy}, ν_{rz}, $\nu_{\theta z}$, $\nu_{\theta r}$ are the Poisson's ratios for orthotropic material; G_{xy} and G_{zr} are the shear moduli.

Thermal Strain

Temperature strain is determined by the coefficients of thermal expansion and difference of temperatures between strained and strainless states. Components of the thermal strain for plane stress and isotropic material are defined by the following equation:

$$\{\varepsilon_0\} = \begin{Bmatrix} \alpha \\ \alpha \\ 0 \end{Bmatrix} \Delta T \; ;$$

plane stress, orthotropic material:

$$\{\varepsilon_0\} = \begin{Bmatrix} \alpha_x \\ \alpha_y \\ 0 \end{Bmatrix} \Delta T \; ;$$

plane strain, isotropic material:

$$\{\varepsilon_0\} = (1+\nu) \begin{Bmatrix} \alpha \\ \alpha \\ 0 \end{Bmatrix} \Delta T \; ;$$

plane strain, orthotropic material:

$$\{\varepsilon_0\} = \begin{Bmatrix} \alpha_x + \nu_{zx}\alpha_z \\ \alpha_y + \nu_{zy}\alpha_z \\ 0 \end{Bmatrix} \Delta T \; ;$$

axisymmetric problem, isotropic material:

$$\{\varepsilon_0\} = \begin{Bmatrix} \alpha \\ \alpha \\ \alpha \\ 0 \end{Bmatrix} \Delta T \; ;$$

axisymmetric problem, orthotropic material:

$$\{\varepsilon_0\} = \begin{Bmatrix} \alpha_z \\ \alpha_r \\ \alpha_\theta \\ 0 \end{Bmatrix} \Delta T ,$$

where α is a coefficient of thermal expansion for isotropic material; α_x, α_y, α_z, α_r, α_θ are the coefficients of thermal expansion along the corresponding axes for orthotropic material; ΔT is the temperature difference between strained and strainless states.

External Forces

QuickField provides way to specify concentrated loads, surface and body forces. The concentrated loads are defined at vertices as two components of the corresponding vector. The surface forces at the edges of the model are specified by the vector components or by the normal pressure. The body forces are defined by their components within blocks of the model. Each component of the body force vector can be specified as a linear function of the coordinates. This feature can be used, for example, to model centrifugal forces. The normal pressure also can be linear function that is useful for hydrostatic pressure.

Note. The concentrated load is specified by the **force per thickness unit** for plane problems and by the **total force value** for axisymmetric ones. In the last case the force can be applied to the point at the axis of symmetry or distributed along the circle around the axis.

Any surface force which is directed along the normal to the surface can be described as a pressure. The pressure is considered positive if it is directed inside region at its outward boundary or from right to left at the inner boundary. Left and right are referred relative to the edge intrinsic direction, which is always counterclockwise for arcs and is determined for line segments by the order of picking vertices when the edge is created.

Restriction Conditions

Rigid constraint condition along one or both axes can be specified at any vertex or along any edge of the model. Prescribed displacement at restrained edge can be specified as a linear function of the coordinates.

Elastic support condition describes a vertex subject to springy force which is proportional to difference between actual and predetermined displacement. The elastic support condition is characterized by the predetermined displacement and the support elasticity.

Note. For problem to be defined correctly the constraint or elastic support conditions have to be specified in such a way to exclude rigid body shifts and rotations of the model or its parts without increasing the potential energy. Two translational and one rotational degrees of freedom have to be restricted for plane problem, in axisymmetric case only shift in z-direction has to be excluded.

Calculated Physical Quantities

For the stress analysis problems the QuickField postprocessor calculates the following set of physical quantities:

- The absolute value of displacement

  $$\delta = \sqrt{\delta_x^2 + \delta_y^2}\text{, or } \delta = \sqrt{\delta_z^2 + \delta_r^2}\ ;$$

- Maximum and minimum principal stresses in the plane of model σ_1 and σ_2;
- Normal and tangential stresses along coordinate axes σ_x, σ_y and τ_{xy} (σ_z, σ_r and τ_{rz} in axisymmetric case);
- Normal stress in out-of-plane direction (σ_z—for xy-plane, σ_θ—for rz-plane). For the plane stress problems this component vanishes by the definition;
- Von Mises criterion (stored energy of deformation):

  $$\sigma_e = \sqrt{\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]}\ ;$$

  where σ_1, σ_2 and σ_3 denote the principal stresses in descending order.
- Tresca criterion (maximum shear):

  $\sigma_e = \sigma_1 - \sigma_3$;
- Mohr-Coulomb criterion:

  $\sigma_e = \sigma_1 - \chi\sigma_3$,

  where

  $$\chi = \frac{[\sigma_+]}{[\sigma_-]},$$

$[\sigma_+]$ and $[\sigma_-]$ denote tensile and compressive allowable stress.

- Drucker-Prager criterion:

$$\sigma_e = \left(1+\sqrt{\chi}\right)\sigma_i - \frac{\sqrt{\chi}-\chi}{1+\sqrt{\chi}}\overline{\sigma} + \frac{1}{[\sigma_-]}\left(\frac{1-\sqrt{\chi}}{1+\sqrt{\chi}}\overline{\sigma}\right)^2,$$

where

$$\sigma_i = \sqrt{\frac{1}{2}\left[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2\right]};$$

$$\overline{\sigma} = \frac{\sigma_1+\sigma_2+\sigma_3}{3}.$$

- Hill failure index for orthotropic materials:

$$\mathrm{F.I.} = \frac{\sigma_1^2}{X_1^2} - \frac{\sigma_1\sigma_2}{X_1^2} + \frac{\sigma_2^2}{X_2^2} + \frac{\tau_{12}^2}{S_{12}^2},$$

where σ_1, σ_2 and τ_{12} are computed stresses in the material directions and,

$X_1 = X_1^T$ if $\sigma_1 > 0$; $X_1 = X_1^C$ if $\sigma_1 < 0$;

$X_2 = X_2^T$ if $\sigma_2 > 0$; $X_2 = X_2^C$ if $\sigma_2 < 0$;

$S_{12} = S_{12}^+$ if $\tau_{12} > 0$; $S_{12} = S_{12}^-$ if $\tau_{12} < 0$,

where X_1^T, X_2^T, X_1^C, X_2^C, S_{12}^+ and S_{12}^- are tensile, compressive and shear allowable stresses.

Coupled Problems

QuickField is capable of importing loads (distributed sources) calculated in some problem into the problem of another type. Following coupling types are supported:

- Heat transfer caused by Joule heat generated in the current flow or time-harmonic electromagnetic problem.
- Thermal stresses based on a calculated temperature distribution.
- Stress analysis of the system loaded by magnetic forces.
- Stresses in electrostatic system.

A special case of coupling allows for importing of the temperature distribution in some steady state or transient heat transfer problem into another transient heat transfer problem as its initial state.

In addition to imported loads, you can define any other loads and boundary conditions, similar to non coupled problem.

You can combine several coupling types in one problem. E.g., after calculating currents distribution, electrostatic and magnetic fields as separate problems based on the same model file, you can calculate temperature distribution from Joule heat and then find stresses caused by temperature and magnetic and electrostatic forces at once. However, such problems are rather rare.

Further we will call the problem, from which the data are being loaded, *the source problem,* and the problem, which imports the data, *the target problem.*

There are several rules to follow with coupled problems:

- Both source and target problem must share a single model file.
- Both problems must use the same formulation (plane or axisymmetric).
- Source problem must be up-to-date when solving the target problem.

Note. In spite of the requirement that both coupled problems must use the same model file, the geometrical region for the problems need not coincide, i.e., some subregions those are in use in one problem, could be excluded from consideration in the other one.

Importing Joule Heat to Heat Transfer Problem

While importing data from current flow problem to heat transfer one, heat sources due to Joule law are assumed in all subregions included into consideration in both source and target problems. In time-harmonic electromagnetic problems Joule heat is generated in all conductors.

Importing Temperature Distribution to Stress Analysis Problem

While calculating thermal stresses, initial strains are assumed in all subregions, which are included into consideration in both problems and possess nonzero value of thermal expansion coefficient (or at least one of its components in anisotropic case).

While importing the temperature distribution from the transient problem, you can choose the moment of time, the state at which you wish to import.

Importing Magnetic Forces to Stress Analysis Problem

While importing magnetic force to stress analysis problem:

- Body force is assumed in all subregions included into consideration in both source and target problems, if those subregions have nonlinear magnetic properties and/or current density is defined (Lorentz force).
- Surface force is assumed at the boundaries separating subregions with different magnetic properties, boundaries with surface current, or outward boundaries in sense of magnetic problem. The surface force is also generated in the cases, when only one subregion, say, to the left of the boundary is active in sense of magnetic problem, and only the subregion to the right of it is active in stress analysis problem.

Importing Electrostatic Forces to Stress Analysis Problem

While importing electrostatic force to stress analysis problem:

- Body force is assumed for all subregions included into consideration in both source and target problems and carrying distributed charge density.
- Surface force is assumed at the boundaries separating subregions with different permittivity, boundaries with surface charge, or outward boundaries in sense of electrostatic problem. The surface force is also generated in the cases, when only one subregion, say, to the left of the boundary is active in sense of electrostatic problem, and only the subregion to the right of it is active in stress analysis problem.

CHAPTER 9

Examples

This chapter contains descriptions of the example problems supplied in the Examples folder. Each problem in this folder is represented by the complete database, which includes geometric model, finite element mesh, definition of material properties, loads and boundary conditions, and ready analysis results. Supplied analysis results allow you to look instantly at the postprocessing capabilities without spending time for preparing data and solving the problem.

QuickField online Help contains detailed step-by-step description of the modeling process, data preparation, and postprocessing of the results for some of the examples described in this chapter. They are provided to illustrate effective modeling techniques and to give you an opportunity to learn QuickField by following an example. See *Tutorial* topic in online Help.

Magnetic Problems

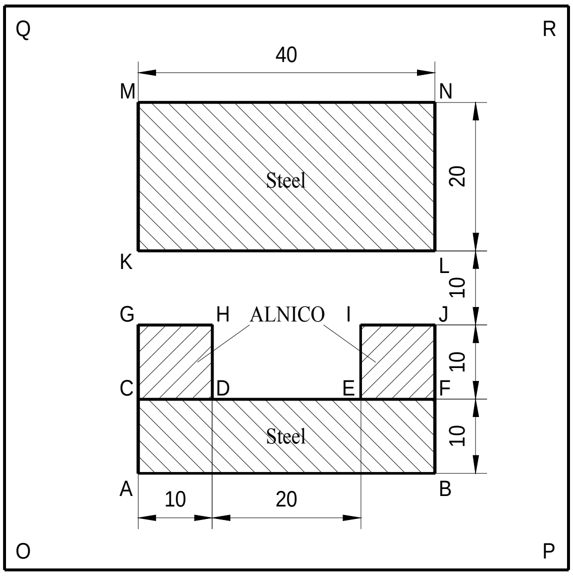
Magn1: Nonlinear Permanent Magnet

A permanent magnet and a steel keeper in the air.

Problem Type:

A nonlinear plane-parallel problem of magnetostatics.

Geometry:



All dimensions are in centimeters.

The permanent magnets are made of ALNICO, coercive force is 147218 A/m. The polarizations of the magnets are along vertical axis opposite to each other. The demagnetization curve for ALNICO:

H (A/m)	-147218	-119400	-99470	-79580	-53710	-19890	0.0
B (T)	0.0	0.24	0.4	0.5	0.6	0.71	0.77

The B-H curve for the steel:

H (A/m)	400	600	800	1000	1400	2000	3000	4000	6000
B (T)	0.73	0.92	1.05	1.15	1.28	1.42	1.52	1.58	1.60

Comparison of Results

Maximum flux density in Y-direction:

ANSYS	0.42
COSMOS/M	0.404
QuickField	0.417

See the Magn1.pbm problem in the Examples folder.

Also see *Tutorial* topic in online Help for detailed step-by-step instructions for this example.

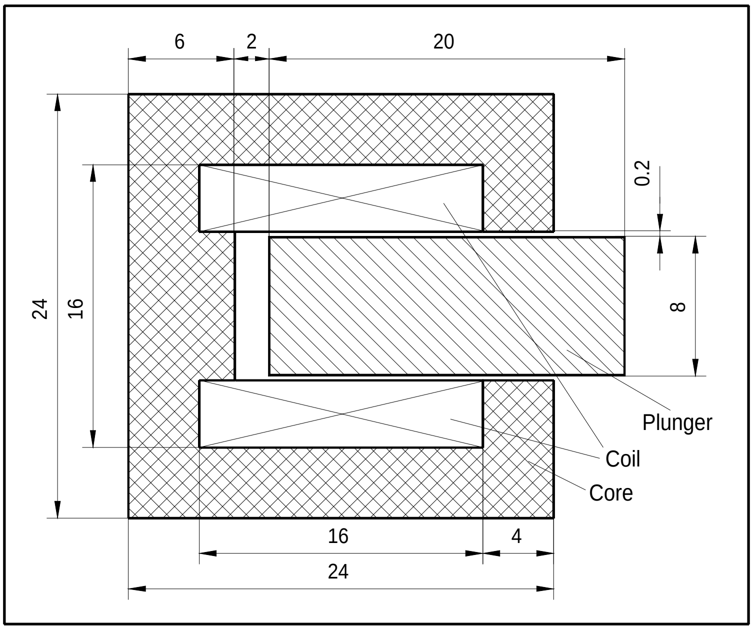
Magn2: Solenoid Actuator

A solenoid actuator consists of a coil enclosed in a ferromagnetic core with a plunger. Calculate the magnetic field and a force applied to the plunger.

Problem Type:

A nonlinear axisymmetric problem of magnetics.

Geometry:



All dimensions are in centimeters.

Given:

Relative permeability of air and coil $\mu = 1$;
Current density in the coil $j = 1 \cdot 10^6$ A/m^2;
The B-H curve for the core and the plunger:

H (A/m)	460	640	720	890	1280	1900	3400	6000
B (T)	0.80	0.95	1.00	1.10	1.25	1.40	1.55	1.65

Problem:

Obtain the magnetic field in the solenoid and a force applied to the plunger.

Solution:

This magnetic system is almost closed, therefore outward boundary of the model can be put relatively close to the solenoid core. A thicker layer of the outside air is included into the model region at the plunger side, since the magnetic field in this area cannot be neglected.

Mesh density is chosen by default, but to improve the mesh distribution, three additional vertices are added to the model. We put one of this vertices at the coil inner surface next to the plunger corner, and two others next to the corner of the core at the both sides of the plunger.

A contour for the force calculation encloses the plunger. It is put in the middle of the air gap between the plunger and the core. While defining the contour of integration, use a strong zoom-in mode to avoid sticking the contour to existing edges.

The calculated force applied to the plunger $F = 374.1$ N.

See the Magn2.pbm problem in the Examples folder.

Comparison of Results

Maximum flux density in Z-direction in the plunger:

	B_z (T)
Reference	0.933
QuickField	1.0183

Reference

D. F. Ostergaard, *"Magnetics for static fields"*, ANSYS revision 4.3, Tutorials, 1987.

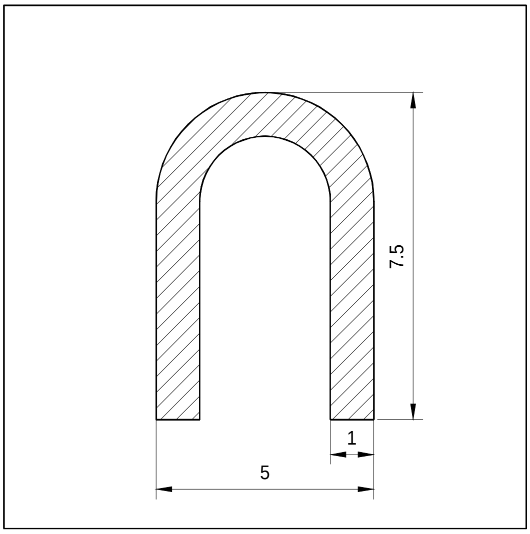
Magn3: Ferromagnetic C-Magnet

A permanent C-magnet in the air. The example demonstrates how to model curved permanent magnet using the equivalent surface currents.

Problem Type:

Plane problem of magnetics.

Geometry of the magnet:



Given:

Relative permeability of the air $\mu = 1$;
Relative permeability of the magnet $\mu = 1000$;
Coercive force of the magnet $H_c = 10000$ A/m.

The polarization of the magnet is along its curvature.

Solution:

To avoid the influence of the boundaries while modeling the unbounded problem, we'll enclose the magnet in a rectangular region of air and specify zero Dirichlet boundary condition on its sides.

Magnetization of straight parts of the magnet is specified in terms of coercive force vector. Effective surface currents simulate magnetization in the middle curved part of the magnet.

See the Magn3.pbm problem in the Examples folder.

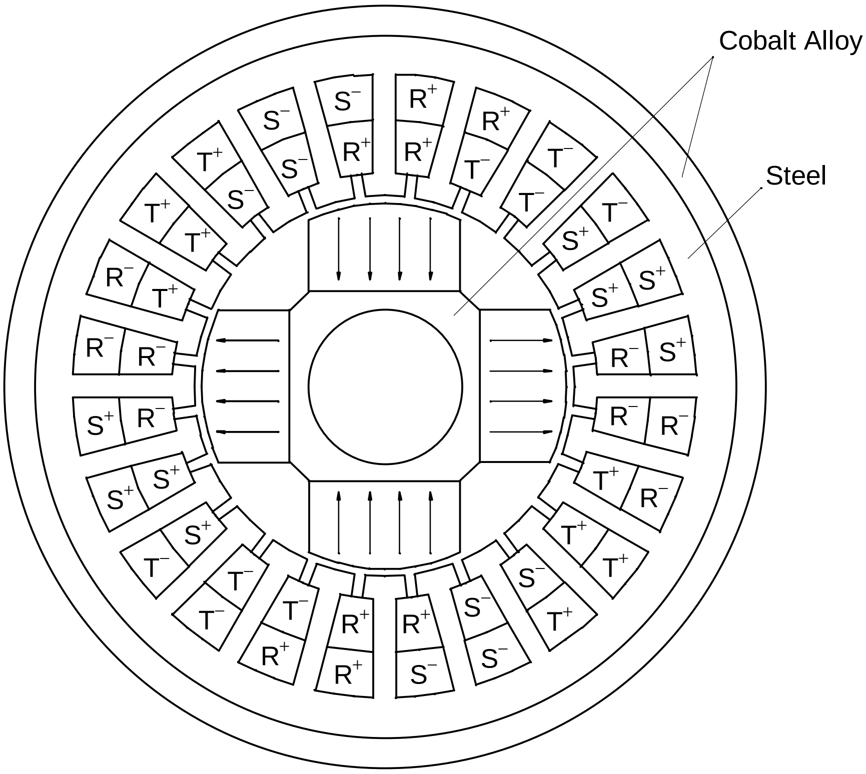
Magn4: Electric Motor

A brushless DC motor with permanent magnets and three phase coil excitation.

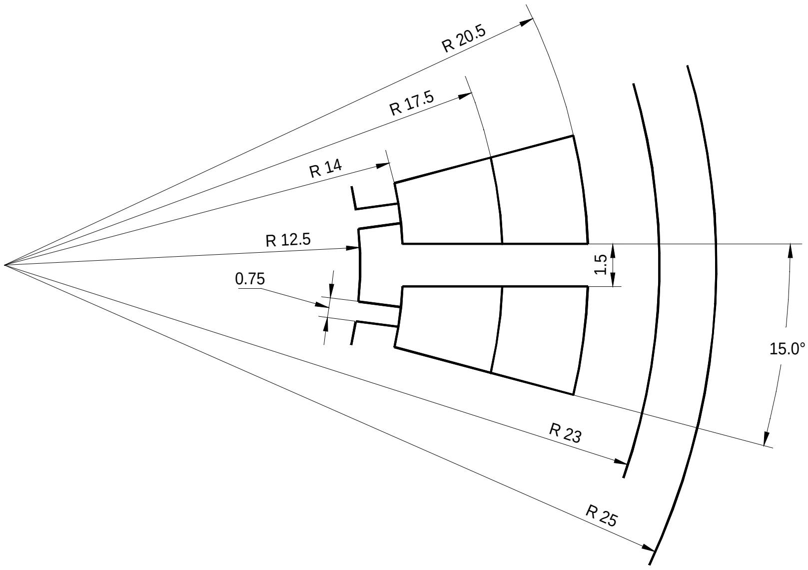
Problem Type:

A nonlinear plane-parallel problem of magnetostatics.

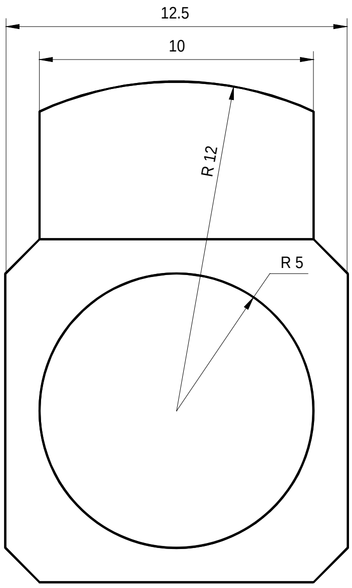
Geometry:



Dimensions of the stator:



Dimensions of the rotor:



All dimensions are in millimeters and degrees Axial length of the motor is 40 mm.

The four magnets are made of Samarium-Cobalt with relative permeability of 1.154 and coercive force of 550000 A/m. The current densities for the coil slots are as follows: $1.3 \cdot 10^6$ A/m^2 on R$^+$, $-1.3 \cdot 10^6$ A/m^2 on R$^-$, $1.3 \cdot 10^6$ A/m^2 on S$^+$, $-1.3 \cdot 10^6$ A/m^2 on S$^-$, and zero on T$^+$ and T$^-$. The inner and outer frames are made of Cobalt-Nickel-Copper-Iron alloy.

The B-H curve for the Cobalt-Nickel-Copper-Iron alloy:

H (A/m)	20	60	80	95	105	120
B (T)	0.19	0.65	0.87	1.04	1.18	1.24
H (A/m)	140	160	180	200	240	2500
B (T)	1.272	1.3	1.32	1.34	1.36	1.45

The B-H curve for the steel:

H (A/m)	400	600	800	1000	1400	2000	3000	4000	6000
B (T)	0.73	0.92	1.05	1.15	1.28	1.42	1.52	1.58	1.60

See the Magn4.pbm problem in the Examples folder.

Also see *Tutorial* topic in online Help for detailed step-by-step instructions for this example.

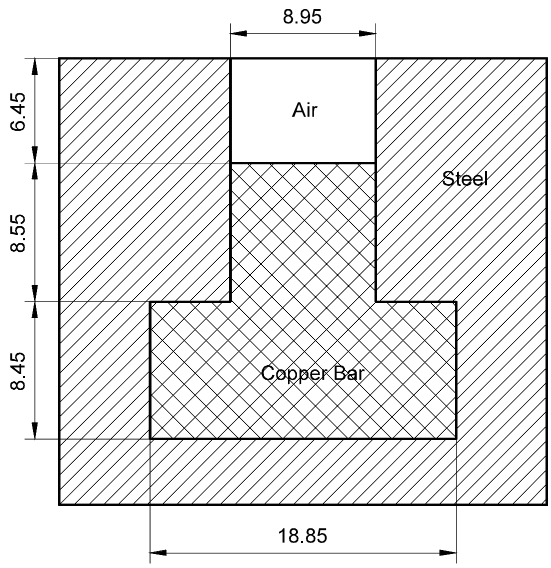
Time-Harmonic Magnetic Problems

HMagn1: Slot Embedded Conductor

Problem Type:

A plane problem of time-harmonic magnetic field.

Geometry:



A solid copper conductor embedded in the slot of an electric machine carries a current *I* at a frequency *f*.

Given:

Magnetic permeability of air $\mu = 1$;
Magnetic permeability of copper $\mu = 1$;
Conductivity of copper $\sigma = 5.8005 \cdot 10^7$ S/m;
Current in the conductor $I = 1$ A;
Frequency $f = 45$ Hz.

Problem:

Determine current distribution within the conductor and complex impedance of the conductor.

Solution:

We assume that the steel slot is infinitely permeable and may be replaced with a Neumann boundary condition. We also assume that the flux is contained within the slot, so we can put a Dirichlet boundary condition along the top of the slot. See HMagn1.pbm problem in the Examples folder for the complete model.

The complex impedance per unit length of the conductor can be obtained from the equation

$$Z = \frac{V}{I},$$

where V is a voltage drop per unit length. This voltage drop can be obtained in the Postprocessor (choose **Results**, **Analyze**, **Values**, **Complex**, and then pick an arbitrary point within the conductor.)

Comparison of Results

	Re Z (Ohm/m)	Im Z (Ohm/m)
Reference	$1.7555 \cdot 10^{-4}$	$4.7113 \cdot 10^{-4}$
QuickField	$1.7550 \cdot 10^{-4}$	$4.7111 \cdot 10^{-4}$

Reference

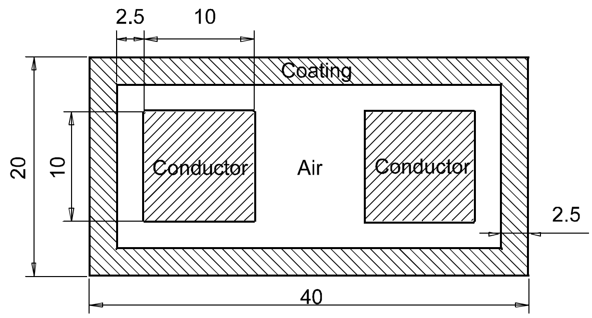
A. Konrad, "*Integrodifferential Finite Element Formulation of Two-Dimensional Steady-State Skin Effect Problems*", IEEE Trans. Magnetics, Vol MAG-18, No. 1, January 1982.

HMagn2: Symmetric Double Line of Conductors

Problem Type:

A plane problem of time-harmonic magnetic field.

Geometry:



Two copper square cross-section conductors with equal but opposite currents are contained inside rectangular ferromagnetic coating. All dimensions are in millimeters.

Given:

Magnetic permeability of air $\mu = 1$;
Magnetic permeability of copper $\mu = 1$;
Conductivity of copper $\sigma = 5.6 \cdot 10^7$ S/m;
Magnetic permeability of coating $\mu = 100$;
Conductivity of copper $\sigma = 1.0 \cdot 10^6$ S/m;
Current in the conductors $I = 1$ A;
Frequency $f = 100$ Hz.

Problem:

Determine current distribution within the conductors and the coating, complex impedance of the line, and power losses in the coating.

Solution:

We assume that the flux is contained within the coating, so we can put a Dirichlet boundary condition on the outer surface of the coating. See HMagn2.pbm problem in the Examples folder for the complete model.

The complex impedance per unit length of the line can be obtained from the equation

$$Z = \frac{V_1 - V_2}{I},$$

where V_1 and V_2 are voltage drops per unit length in each conductor. These voltage drops are equal with opposite signs due to the symmetry of the model. To obtain an impedance choose **Impedance Wizard** from **View** menu or double click it in the calculator tree and then seelct both **Conductor 1** and **Conductor 2** items in the conductors list.

The impedance of the line $Z = 4.87 \cdot 10^{-4} + i\, 7.36 \cdot 10^{-4}$ Ohm/m.

To obtain power losses in the coating:

1. In the postprocessing mode, choose **Integral Values** from **View** menu. Then switch on contour editing mode using **Pick Elements** command from **Contour** menu and pick the coating block to create the contour.
2. Double click on **Joule heat** item in the list of integral quantities or click the gray button to the left of it.

The power losses in the coating $P = 4.28 \cdot 10^{-5}$ W/m.

Electrostatic Problems

Elec1: Microstrip Transmission Line

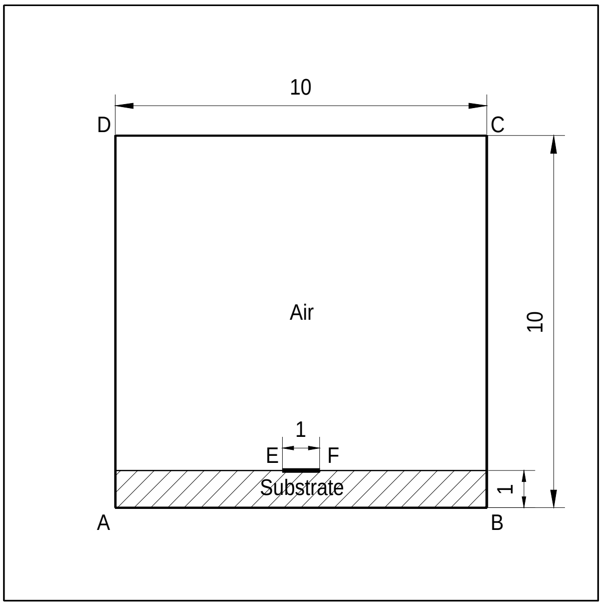
A shielded microstrip transmission line consists of a substrate, a microstrip, and a shield.

Problem Type:

Plane-parallel problem of electrostatics.

Geometry:

The transmission line is directed along *z*-axis, its cross section is shown on the sketch. The rectangle *ABCD* is a section of the shield, the line *EF* represents a conductor strip.



Given:

Relative permittivity of air ε = 1;
Relative permittivity of substrate ε = 10.

Problem:

Determine the capacitance of a transmission line.

Solution:

There are several different approaches to calculate the capacitance of the line:

- To apply some distinct potentials to the shield and the strip and to calculate the charge that arises on the strip;
- To apply zero potential to the shield and to describe the strip as having constant but unknown potential and carrying the charge, and then to measure the potential that arises on the strip.

Both these approaches make use of the equation for capacitance:

$$C = \frac{q}{U}.$$

Other possible approaches are based on calculation of stored energy of electric field. When the voltage is known:

$$C = \frac{2W}{U^2},$$

and when the charge is known:

$$C = \frac{q^2}{2W}$$

Experiment with this example shows that energy-based approaches give little bit less accuracy than approaches based on charge and voltage only. The first approach needs to get the charge as a value of integral along some contour, and the second one uses only a local value of potential, this approach is the simplest and in many cases the most reliable.

The first and third approaches are illustrated in the Elec1_1.pbm problem in the Examples folder, and the Elec1_2.pbm explains the second and the fourth approaches.

Results:

Theoretical result: C = 178.1 pF/m.

Approach 1: C = 177.83 pF/m (99.8%).

Approach 2: C = 178.47 pF/m (100.2%).

Approach 3: C = 177.33 pF/m (99.6%).

Approach 4: C = 179.61 pF/m (100.8%).

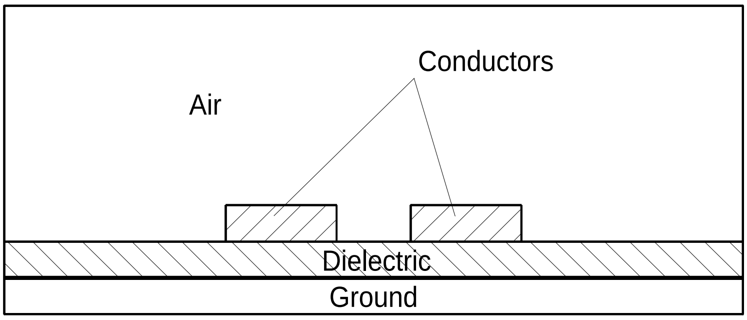
See *Tutorial* topic in online Help for detailed step-by-step instructions for this example.

Elec2: Two Conductor Transmission Line

Problem Type:

A plane problem of electrostatics.

Geometry:



The problem's region is bounded by ground from the bottom side and extended to infinity on other three sides.

Given:

Relative permittivity of air ε = 1;
Relative permittivity of dielectric ε = 2.

Problem:

Determine self and mutual capacitance of conductors.

Solution:

To avoid the influence of outer boundaries, we'll define the region as a rectangle large enough to neglect side effects. To calculate the capacitance matrix we set the voltage U = 1 V on one conductor and U = 0 on the another one.

Self capacitance: $C_{11} = C_{22} = \dfrac{Q_1}{U_1}$;

Mutual capacitance: $C_{12} = C_{21} = \dfrac{Q_2}{U_1}$;

where charge Q_1 and Q_2 are evaluated on rectangular contours around conductor 1 and 2 away from their edges. We chose the contours for the C_{11} and C_{12} calculation to be rectangles $-6 \le x \le 0$, $0 \le y \le 4$ and $0 \le x \le 6$, $0 \le y \le 4$ respectively.

Comparison of Results

	C_{11} (F/m)	C_{12} (F/m)
Reference	$9.23 \cdot 10^{-11}$	$-8.50 \cdot 10^{-12}$
QuickField	$9.43 \cdot 10^{-11}$	$-8.57 \cdot 10^{-12}$

Reference

A. Khebir, A. B. Kouki, and R. Mittra, *"An Absorbing Boundary Condition for Quasi-TEM Analysis of Microwave Transmission Lines via the Finite Element Method"*, Journal of Electromagnetic Waves and Applications, 1990.

See the Elec2.pbm problem in the Examples folder.

Steady State Heat Transfer Problems

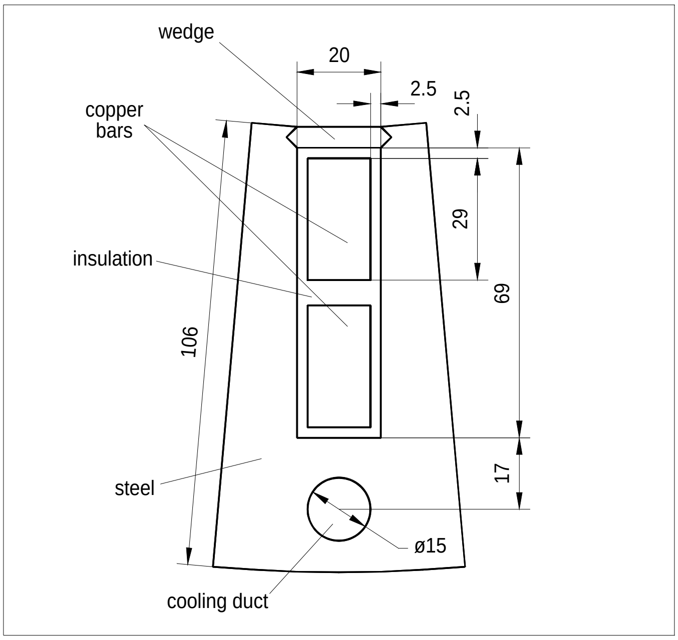
Heat1: Slot of an Electric Machine

Temperature field in the stator tooth zone of power synchronous electric machine.

Problem Type:

The plane-parallel problem of heat transfer with convection.

Geometry:



All dimensions are in millimeters. Stator outer diameter is 690 mm. Domain is a 10-degree segment of stator transverse section. Two armature bars laying in the slot release ohmic loss. Cooling is provided by convection to the axial cooling duct and both surfaces of the core.

Given:

Specific copper loss: 360000 W/m^3;
Heat conductivity of steel: 25 J/K·m;
Heat conductivity of copper: 380 J/K·m;
Heat conductivity of insulation: 0.15 J/K·m;
Heat conductivity of wedge: 0.25 J/K·m;

Inner stator surface:

Convection coefficient: 250 W/K·m^2;
Temperature of contacting air: 40°C.

Outer stator surface:

Convection coefficient: 70 W/K·m^2;
Temperature of contacting air: 20°C.

Cooling duct:

Convection coefficient: 150 W/K·m^2;
Temperature of contacting air: 40°C.

See the Heat1.pbm problem in the Examples folder.

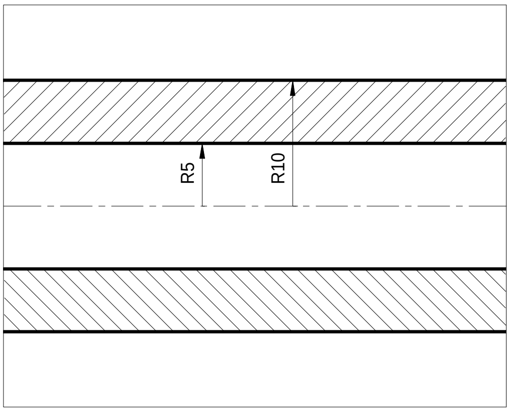
Heat2: Cylinder with Temperature Dependent Conductivity

A very long cylinder (infinite length) is maintained at temperature T_i along its internal surface and T_o along its external surface. The thermal conductivity of the cylinder is known to vary with temperature according to the linear function $\lambda(T) = C_0 + C_1 \cdot T$.

Problem Type:

An axisymmetric problem of nonlinear heat transfer.

Geometry:



Given:

R_i = 5 mm, R_o = 10 mm;
T_i = 100°C, T_o = 0°C;
C_0 = 50 W/K·m, C_1 = 0.5 W/K·m.

Problem:

Determine the temperature distribution in the cylinder.

Solution:

The axial length of the model is arbitrarily chosen to be 5 mm.

Comparison of Results

Radius	QuickField	Theory
0.6	79.2	79.2
0.7	59.5	59.6
0.8	40.2	40.2
0.9	20.7	20.8

See the Heat2.pbm problem in the Examples folder.

Transient Heat Transfer Problems

THeat1: Heating and Cooling of a Slot of an Electric Machine

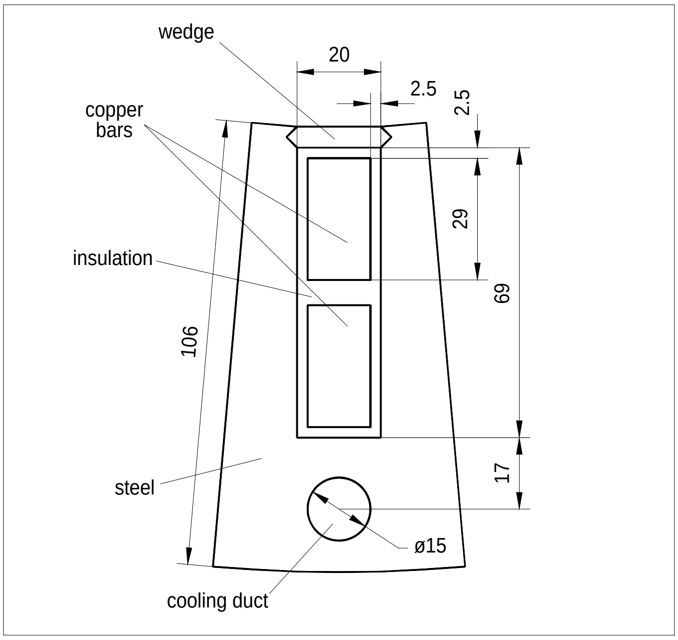
Changing temperature field in the stator tooth zone of power synchronous electric motor during a loading-unloading cycle.

Problem Type:

The plane-parallel problem of transient heat transfer with convection.

Geometry:

The same as with Heat1 example:



Given:

1. Working Cycle

We assume the uniformly distributed temperature before the motor was suddenly loaded. The cooling conditions supposed to be constant during the heating process. We keep track of the temperature distribution until it gets almost steady-state. Then we start to solve the second problem –cooling of the suddenly stopped motor. The initial temperature field is imported from the previous solution. The cooling condition supposed constant, but different from those while the motor was being loaded.

2. Material Properties

	Heat Conductivity (J/K·m)	Specific Heat (J/K·m)	Mass Density (kg/m^3)
Steel Core	25	465	7833
Copper Bar	380	380	8950
Bar Insulation	0.15	1800	1300
Wedge	0.25	1500	1400

3. Heat Sources and Cooling Conditions

	Loading		**Stopped**	
Initial Temperature				
The entire model	0 (°C)		As calculated at the end of loading phase	
Heat Sources				
Specific power loss in copper bars (W/m^3)	360000		0	
Cooling Conditions				
	Convection coefficient (W/K·m^2)	Temperature of contacting air (°C)	Convection coefficient (W/K·m^2)	Temperature of contacting air (°C)
Inner stator surface	250	40	20	40
Outer stator surface	70	20	70	20
Cooling duct	150	40	20	40

Solition:

Each phase of the loading cycle is modeled by a separate QuickField problem. For the loading phase the initial temperature is set to zero, for the cooling phase the initial thermal distribution is imported from the final time moment of the previous solution.

Moreover, we decide to break the cooling phase into two separate phases. For the first phase we choose time step as small as 100 s, because the rate of temperature change is relatively high. This allows us to see that the temperature at the slot bottom first increases by approximately 1 grad for 300 seconds, and then begins decreasing. The second stage of cooling, after 1200 s, is characterized by relatively low rate of temperature changing. So, we choose for this phase the time step to be 600 s.

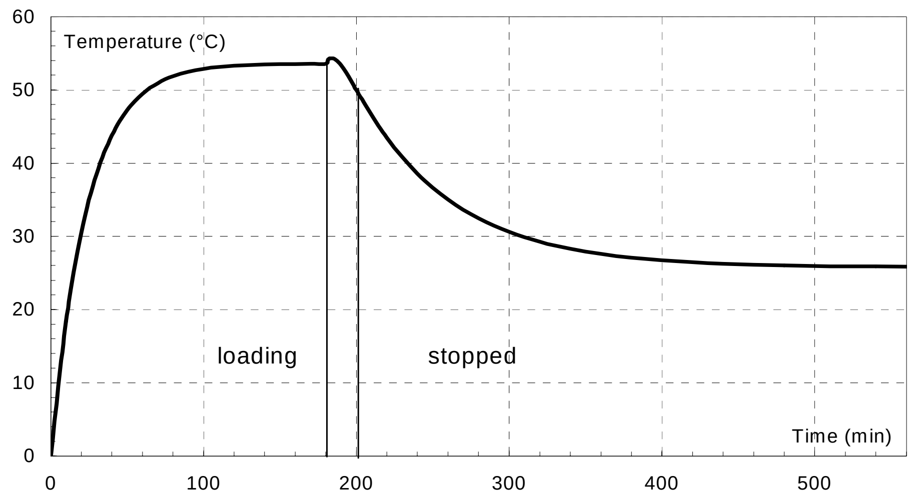
For the heating process, the time step of 300 sec is chosen.

Please see following problems in the Examples folder:

- THeat1Ld.pbm for loading phase, and
- THeat1S1.pbm for the beginning of stopped phase, and
- THeat1S2.pbm for the end of stopped phase

Results:

Temperature vs. time dependence at the bottom of the slot (where a temperature sensor usually is placed).



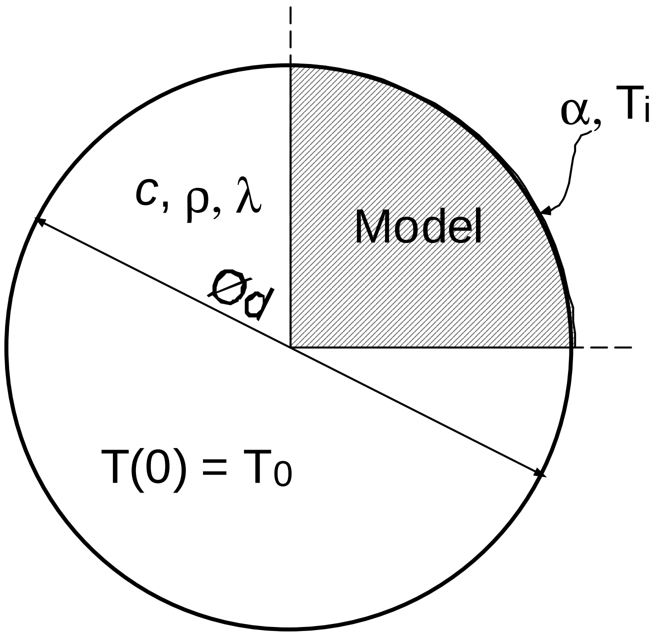
THeat2: Temperature Response of a Suddenly Cooled Wire

Determine the temperature response of a copper wire of diameter *d*, originally at temperature T_o, when suddenly immersed in air at temperature T_i. The convection coefficient between the wire and the air is α.

Problem Type:

A plane-parallel problem of transient heat transfer with convection.

Geometry:



Given:

d = 0.015625 in;
T_i = 37.77°C, T_o = 148.88°C;
C = 380.16 J/kg·K, ρ = 8966.04 kg/m^3;
α = 11.37 W/K·m^2.

Problem:

Determine the temperature in the wire.

Solution:

To set the non-zero initial temperature we have to solve an auxiliary steady state problem, whose solution is uniform distribution of the temperature T_0

The final time of 180 sec is sufficient for the theoretical response comparison. A time step of 4.5 sec is used.

Comparison of Results

	Temperature, C		
Time	**QuickField**	**ANSYS**	**Reference**
45 sec	91.37	91.38	89.6
117 sec	54.46	54.47	53.33
180 sec	43.79	43.79	43.17

See the THeat2.pbm (main) and THeat2_i.pbm (auxiliary) problems in the Examples folder.

Reference

Kreif F., *"Principles of Heat Transfer"*, International Textbook Co., Scranton, Pennsylvania, 2nd Printing, 1959, Page 120, Example 4-1.

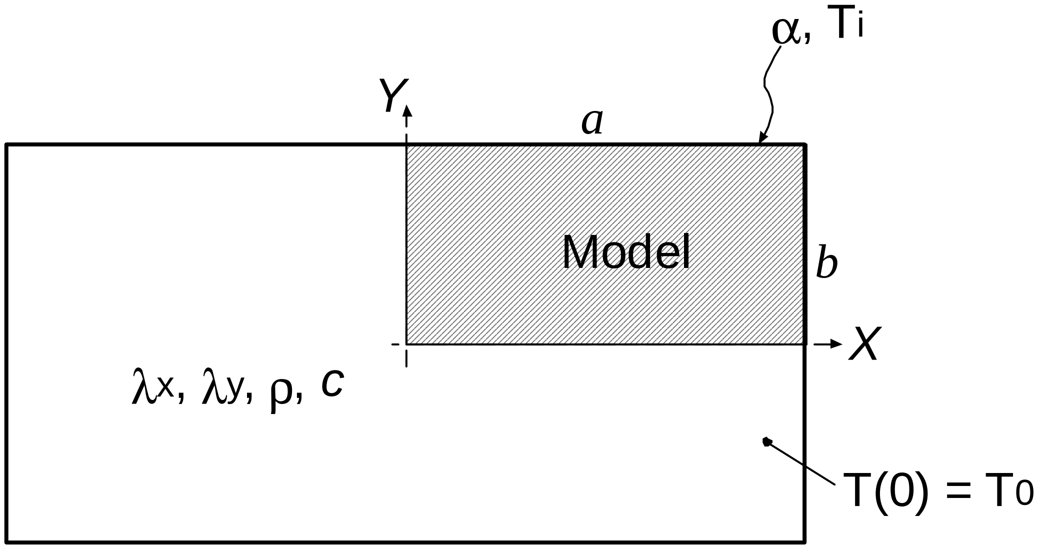
THeat3: Transient Temperature Distribution in an Orthotropic Metal Bar

A long metal bar of rectangular cross-section is initially at a temperature T_o and is then suddenly quenched in a large volume of fluid at temperature T_i. The material conductivity is orthotropic, having different X and Y directional properties. The surface convection coefficient between the wire and the air is α.

Problem Type:

A plane-parallel problem of transient heat transfer with convection.

Geometry:



Given:

a = 2 in, b = 1 in

λ_x = 34.6147 W/K·m, λ_y = 6.2369 W/K·m;

T_i = 37.78°C, T_o = 260°C;

α = 1361.7 W/K·m^2;

C = 37.688 J/kg·K, ρ = 6407.04 kg/m^3.

Problem:

Determine the temperature distribution in the slab after 3 seconds at the center, corner edge and face centers of the bar.

Solution:

To set the non-zero initial temperature we have to solve an auxiliary steady state problem, whose solution is uniform distribution of the temperature T_0

A time step of 0.1 sec is used.

Comparison of Results

	Temperature, C		
Point	**QuickField**	**ANSYS**	**Reference**
(0,0) in	238.7	239.4	237.2
(2,1) in	66.43	67.78	66.1
(2,0) in	141.2	140.6	137.2
(0,1) in	93.8	93.3	94.4

See the THeat3.pbm (main) and THeat3_i.pbm (auxiliary) problem in the Examples folder.

Reference

Schneider P.J., *"Conduction Heat Transfer"*, Addison-Wesley Publishing Co., Inc, Reading, Mass., 2nd Printing, 1957, Page 261, Example 10-7.

Stress Analysis Problems

Stres1: Perforated Plate

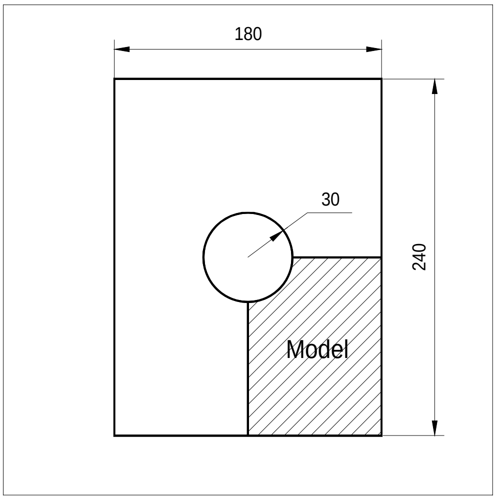
A thin rectangular sheet with a central hole subject to tensile loading.

Problem Type:

Plane problem of stress analysis (plane stress formulation).

Geometry of the plate:

Length: 240 mm;
Width: 180 mm;
Radius of central opening: 30 mm;
Thickness: 5 mm.



Given:

Young's modulus E = 207000 N/mm^2;
Poisson's ratio ν = 0.3.

The uniform tensile loading (40 N/mm^2) is applied to the bottom edge of the structure.

Problem:

Determine the concentration factor due to presence of the central opening.

Solution:

Due to mirror symmetry one quarter of the structure is presented, and internal boundaries are restrained in *X* and *Y* directions respectively.

The concentration factor may be obtained from the loading stress (40 N/mm^2) and the maximum computed stress (146 N/mm^2) as

$k = 146 / 40 = 3.65.$

See the Stres1.pbm problem in the Examples folder.

Coupled Problems

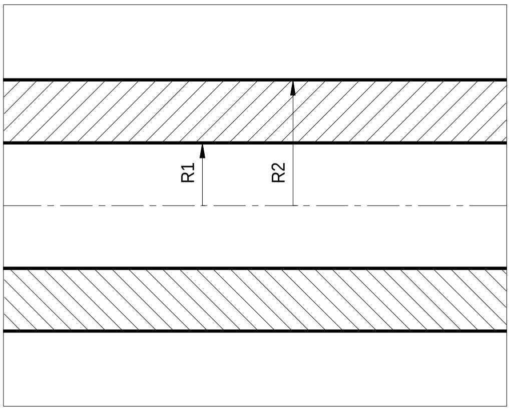
Coupl1: Stress Distribution in a Long Solenoid

A very long, thick solenoid has an uniform distribution of circumferential current. The magnetic flux density and stress distribution in the solenoid has to be calculated.

Problem Type:

An axisymmetric problem of magneto-structural coupling.

Geometry:



Given:

Dimensions $R_i = 1$ cm, $R_o = 2$ cm;
Relative permeability of air and coil $\mu = 1$;
Current density $j = 1 \cdot 10^5$ A/m^2;
Young's modulus $E = 1.075 \cdot 10^{11}$ N/m^2;
Poisson's ratio $\nu = 0.33$.

Problem:

Calculate the magnetic flux density and stress distribution.

Solution:

Since none of physical quantities varies along z-axis, a thin slice of the solenoid could be modeled. The axial length of the model is arbitrarily chosen to be 0.2 cm. Radial component of the flux density is set equal to zero at the outward surface of the solenoid. Axial displacement is set equal to zero at the side edges of the model to reflect the infinite length of the solenoid.

Comparison of Results

Magnetic flux density and circumferential stress at r = 1.3 cm:

	B_z (T)	σ_θ (N/m^2)
Reference	$8.796 \cdot 10^{-3}$	97.407
QuickField	$8.798 \cdot 10^{-3}$	96.71

Reference

F. A. Moon, *"Magneto-Solid Mechanics"*, John Wiley & Sons, N.Y., 1984, Chapter 4.

See the Coupl1MS.pbm and Coupl1SA.pbm problems in the Examples folder for magnetic and structural parts of this problem respectively.

Also see *Tutorial* topic in online Help for detailed step-by-step instructions for this example.

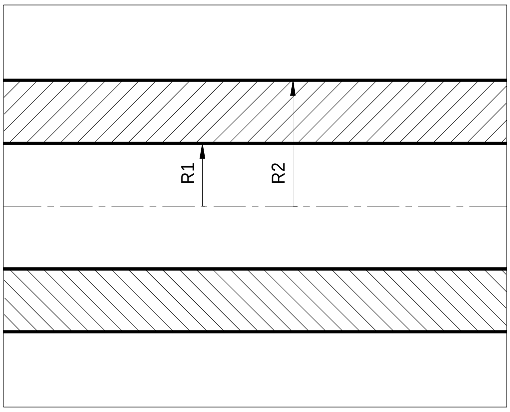
Coupl2: Cylinder Subject to Temperature and Pressure

A very long, thick-walled cylinder is subjected to an internal pressure and a steady state temperature distribution with T_i and T_o temperatures at inner and outer surfaces respectively. Calculate the stress distribution in the cylinder.

Problem Type:

An axisymmetric problem of thermal-structural coupling.

Geometry:



Given:

Dimensions R_i = 1 cm, R_o = 2 cm;
Inner surface temperature T_i = 100°C;
Outer surface temperature T_o = 0°C;
Coefficient of thermal expansion $\alpha = 1 \cdot 10^{-6}$ 1/K;
Internal pressure $P = 1 \cdot 10^{6}$ N/m^2;
Young's modulus $E = 3 \cdot 10^{11}$ N/m^2;
Poisson's ratio $\nu = 0.3$.

Problem:

Calculate the stress distribution.

Solution:

Since none of physical quantities varies along z-axis, a thin slice of the cylinder can be modeled. The axial length of the model is arbitrarily chosen to be 0.2 cm. Axial displacement is set equal to zero at the side edges of the model to reflect the infinite length of the cylinder.

Comparison of Results

Radial and circumferential stress at r = 1.2875 cm:

	σ_r (N/m^2)	σ_θ (N/m^2)
Theory	$-3.9834 \cdot 10^6$	$-5.9247 \cdot 10^6$
QuickField	$-3.959 \cdot 10^6$	$-5.924 \cdot 10^6$

Reference

S. P. Timoshenko and Goodier, *"Theory of Elasticity"*, McGraw-Hill Book Co., N.Y., 1961, pp. 448-449.

See the Coupl2HT.pbm and Coupl2SA.pbm problems in the Examples folder for the corresponding heat transfer and structural parts of this problem.

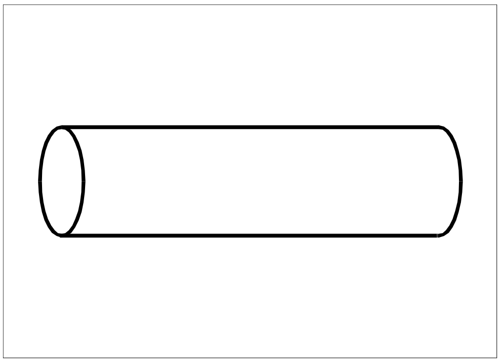
Coupl3: Temperature Distribution in an Electric Wire

Calculate the temperature distribution in a long current carrying wire.

Problem Type:

An axisymmetric problem of electro-thermal coupling.

Geometry:



Given:

Wire diameter $d = 10$ mm;
Resistance $R = 3{\cdot}10^{-4}$ Ω/m;
Electric current $I = 1000$ A;
Thermal conductivity $\lambda = 20$ W/K·m;
Convection coefficient $\alpha = 800$ W/K·m^2;
Ambient temperature $T_o = 20$°C.

Problem:

Calculate the temperature distribution in the wire.

Solution:

We arbitrary chose a 10 mm piece of wire to be represented by the model. For data input we need the wire radius $r = 5$ mm, and the resistivity of material:

$$\rho = \frac{\pi d^2 R}{4} = 2.356 \cdot 10^{-8}\ (\Omega \cdot \mathrm{m}),$$

and voltage drop for our 10 mm piece of the wire:

$$\Delta U = I \cdot R \cdot l = 3 \cdot 10^{-3}\ (\mathrm{V}).$$

For the current flow problem we specify two different voltages at two sections of the wire, and a zero current condition at its surface. For heat transfer problem we specify zero flux conditions at the sections of the wire and a convection boundary condition at its surface.

Comparison of Results

Center line temperature:

	T (°C)
Theory	33.13
QuickField	33.14

Reference

W. Rohsenow and H. Y. Choi, *"Heat, Mass, and Momentum Transfer"*, Prentice-Hall, N.J., 1963.

See the Coupl3CF.pbm and Coupl3HT.pbm problems in the Examples folder for the corresponding current flow and heat transfer parts of this problem.

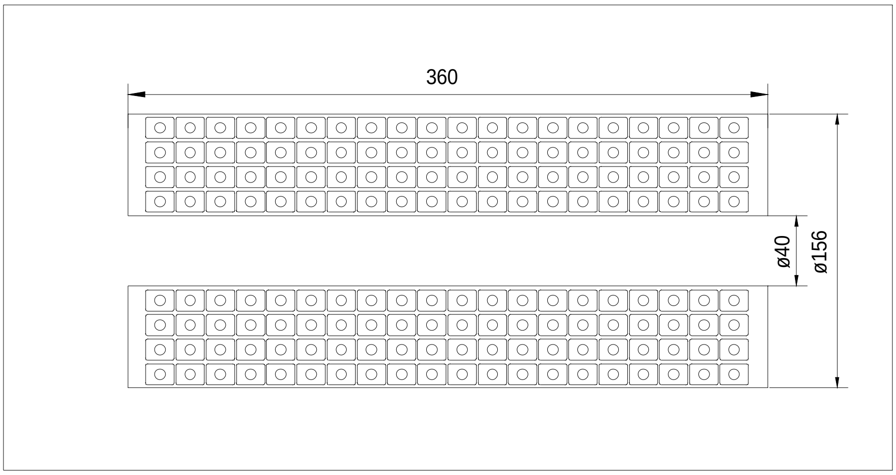
Coupl4: Tokamak Solenoid

The central solenoid of the ohmic heating system for a tokamak fusion device.

Problem Type:

An axisymmetric problem of magneto-structural coupling.

Geometry:



The solenoid consists of 80 superconducting coils fixed in common plastic structure. Due to mirror symmetry one half of the structure is modeled.

Given:

Data for magnetic analysis:

Current density in coils $j = 3{\cdot}10^8$ A/m^2;
Magnetic permittivity of plastic, coils and liquid helium inside coils $\mu = 1$.

Data for stress analysis:

Copper of coils:

Young's modulus $E = 7.74{\cdot}10^{10}$ N/m^2;
Poisson's ratio $\nu = 0.335$;
Maximum allowable stress: $2.2{\cdot}10^8$ N/m^2.

Plastic structure:

Young's modulus $E = 2{\cdot}10^{11}$ N/m^2;
Poisson's ratio $\nu = 0.35$;
Maximum allowable stress: $1{\cdot}10^9$ N/m^2.

In the Examples folder the Coupl4MS.pbm is the problem of calculating the magnetic field generated by the solenoid, and Coupl4SA.pbm analyzes stresses and deformations in coils and plastic structure due to Lorentz forces acting on the coils.

Index

W

X

Y

Z