

 **CST PARTICLE STUDIO**

**Workflow &
Solver Overview**

Copyright

© CST 2005–2015

CST – Computer Simulation Technology AG

All rights reserved.

Information in this document is subject to change without notice. The software described in this document is furnished under a license agreement or non-disclosure agreement. The software may be used only in accordance with the terms of those agreements.



No part of this documentation may be reproduced, stored in a retrieval system, or transmitted in any form or any means electronic or mechanical, including photocopying and recording, for any purpose other than the purchaser's personal use without the written permission of CST.

Trademarks

CST, CST STUDIO SUITE, CST MICROWAVE STUDIO, CST EM STUDIO, CST PARTICLE STUDIO, CST CABLE STUDIO, CST PCB STUDIO, CST MPHYSICS STUDIO, CST MICROSTRIPES, CST DESIGN STUDIO, CST BOARDCHECK, PERFECT BOUNDARY APPROXIMATION (PBA), and the CST logo are trademarks or registered trademarks of CST in North America, the European Union, and other countries. Other brands and their products are trademarks or registered trademarks of their respective holders and should be noted as such.

CST – Computer Simulation Technology AG
www.cst.com

Contents

CHAPTER 1 — INTRODUCTION	3
Welcome.....	3
How to Get Started Quickly	3
What is CST PARTICLE STUDIO?.....	3
Who Uses CST PARTICLE STUDIO?.....	4
CST PARTICLE STUDIO Key Features	5
General.....	5
Structure Modeling.....	5
Particle Tracking Simulator.....	6
Particle-in-Cell Simulator.....	6
Wakefield Simulator.....	7
Eigenmode Simulator	8
Electrostatics Simulator	9
Magnetostatics Simulator	9
Visualization and Secondary Result Calculation.....	9
Result Export.....	10
Automation	10
About This Manual.....	11
Document Conventions	11
Your Feedback	11
CHAPTER 2 — SIMULATION WORKFLOWS.....	12
Simulation Workflow: Particle Tracking	13
The Structure	13
Create a New Project.....	14
Open the Tracking QuickStart Guide	16
Define the Units	17
Define the Background Material.....	17
Model the Structure	17
Define Potentials and Magnets	24
Visualize and Refine the Mesh	27
Define Particle Sources	29
Define Boundary Conditions	33
Start the Simulation	35
Analyze the Results	37
Parameterization of the Model	40
Automatic Optimization of the Structure	44
Additional Information: The Particle Tracking Specials Dialog Box	47
Summary	48
Simulation Workflow: Particle-in-Cell	49
The Structure	49
Create a New Project.....	50
Open the PIC QuickStart Guide	52
Define the Units	53
Define the Background Material.....	53
Model the Structure	53
Define the Particle Source	59
Simulation Setup.....	64
Refine the mesh.....	66
Define Particle Monitors.....	67
Start the Simulation	68
Analyze the Simulation Results	69
Additional information: Interaction of particles with materials	73

Additional information: Acceleration Features	74
Summary	74
Simulation Workflow: Wakefield	75
The Structure	75
Create a New Project.....	76
Open the Wakefield QuickStart Guide	78
Define the Units	78
Define the Background Material.....	79
Model the Structure	79
Define the Particle Beam Source.....	83
Define Boundary and Symmetry Conditions	84
Visualize the Mesh.....	86
Define a 2D Time Domain Field Monitor	88
Start the Simulation	89
Analyze the Simulation Results	90
Additional Information: Wake Postprocessor	93
Summary	95
CHAPTER 3 – COUPLED SIMULATIONS	96
Considering Electromagnetic Fields	96
Computation of Electromagnetic Fields	96
Definition of Analytic Magnetic Fields	97
Import of Electromagnetic Fields	98
Particle Interfaces.....	99
Export of Thermal Losses.....	100
CHAPTER 4 — FIND FURTHER INFORMATION.....	101
The QuickStart Guide	101
Online Documentation	102
Examples.....	102
Technical Support.....	102
History of Changes	103

Chapter 1 — Introduction

Welcome

Welcome to CST PARTICLE STUDIO, the powerful and easy-to-use electromagnetic field and charged particle dynamics simulation software. This program combines a user-friendly interface with high simulation performance.

CST PARTICLE STUDIO is part of the CST STUDIO SUITE. Please refer to the *CST STUDIO SUITE Getting Started* manual first. The following explanations assume that you have already installed the software and familiarized yourself with the basic concepts of the user interface.

How to Get Started Quickly

We recommend you proceed as follows:

1. Read the *CST STUDIO SUITE Getting Started* manual.
2. Work through this document carefully. It provides all the basic information necessary to understand the advanced documentation.
3. Work through the online help system's tutorials by choosing the example which best suits your needs.
4. Look at the examples folder in the installation directory. The different application types will give you a good impression of what has already been done with the software. Please note that these examples are designed to give you a basic insight into a particular application domain. Real-world applications are typically much more complex and harder to understand if you are not familiar with the basic concepts.
5. Start with your own first example. Choose a reasonably simple example which will allow you to become familiar with the software quickly.
6. After you have worked through your first example, contact technical support for hints on possible improvements to achieve even more efficient usage of CST PARTICLE STUDIO.

What is CST PARTICLE STUDIO?

CST PARTICLE STUDIO is a fully featured software package for the design and analysis of electromagnetic components for accelerating and guiding charged particle beams. It simplifies the structure generation by providing a powerful solid modeling front end based on the industry-standard ACIS modeling kernel. Strong graphic feedback simplifies the definition of your device even further. After the component is modeled, a fully automatic meshing procedure (based on an expert system) is applied for the electromagnetic computation before the simulation engine is started.

The simulators support the Perfect Boundary Approximation (PBA) feature, which increases the accuracy of the electromagnetic simulation significantly in comparison to conventional simulators. To calculate electromagnetic fields and analyze particle

dynamics this software contains three different solvers: a time domain **Wakefield simulator**, a time domain **Particle-in-Cell solver** and a **Particle Tracking solver**.

Additionally, CST MPHYSICS STUDIO allows a subsequent thermal or mechanical analysis.

If you are unsure which solver best suits your needs, contact your local sales office for further assistance.

Each solver's simulation results can be visualized with a variety of different options. Again, a strongly interactive interface will help you to achieve the desired insight into your device quickly.

The last – but not least – outstanding feature is the full parameterization of the structure modeler, which enables the use of variables in the definition of your component. In combination with the built-in optimizer and parameter sweep tools, CST PARTICLE STUDIO is capable of both the analysis and design of particle accelerating devices.

Who Uses CST PARTICLE STUDIO?

Anyone who has to deal with electromagnetic problems that involve the effect of charged particle dynamics will greatly benefit from using CST PARTICLE STUDIO. The program is especially suited to the fast, efficient analysis and design of components like electron guns, deflecting devices, guiding configurations and more. Since the underlying method is a general 3D approach, CST PARTICLE STUDIO can solve virtually any field problem that involves interaction with charged particles.

The software is based on an electromagnetic solving method which requires the discretization of the entire calculation volume; for this reason the applications are limited only by the complexity of the structure.

CST PARTICLE STUDIO Key Features

The following list gives you an overview of CST PARTICLE STUDIO's main features. Please note that not all of these features may be available to you because of license restrictions. Please contact a sales office for more information.

General

- Native graphical user interface based on Windows 7, Windows 2008 Server R2, Windows 8, Windows 2012 Server, Windows 8.1 or Windows 2012 Server R2
- The structure can be viewed either as a 3D model or as a schematic. The latter allows a parametrized approach of coupled simulation with our System Assembly and Modeling workflow.
- Various independent solver strategies allow accurate results with a high performance
- For specific solvers, highly advanced numerical techniques offer features like Perfect Boundary Approximation (PBA)® for hexahedral grids and curved and higher order elements for tetrahedral meshes

Structure Modeling

- Advanced ACIS¹-based, parametric solid modeling front end with excellent structure visualization
- Feature-based hybrid modeler allows quick structural changes
- Import of 3D CAD data from ACIS SAT (e.g. AutoCAD®, ACIS SAB, Autodesk Inventor®, IGES, VDA-FS, STEP, Pro/ENGINEER®, CATIA 4®, CATIA 5®, Siemens NX, Parasolid, Solid Edge, SolidWorks, CoventorWare®, Mecadtron®, NASTRAN, STL or OBJ files)
- Import of 2D CAD data from DXF™, GDSII and Gerber RS274X, RS274D files
- Import of EDA data from design flows including Cadence Allegro® / APD® / SIP®, Mentor Graphics Expedition®, Mentor Graphics PADS®, Mentor Graphics HyperLynx®, Zuken CR-5000® / CR-8000® and ODB++® (e.g. Mentor Graphics Boardstation®, CADSTAR®, Visula®)
- Import of PCB designs originating from CST PCB STUDIO®
- Import of 2D and 3D sub models
- Import of Agilent ADS® layouts
- Import of Sonnet® EM models
- Import of a visible human model dataset or other voxel datasets
- Export of CAD data to ACIS SAT, ACIS SAB, IGES, STEP, NASTRAN, STL, DXF™, GDSII, Gerber or POV files
- Parameterization for imported CAD files
- Material database
- Structure templates for simplified problem setup

¹ Portions of this software are owned by Spatial Corp. © 1986 – 2014. All Rights Reserved.

Particle Tracking Simulator

- Arbitrary shaped particle source surfaces
- Circular particle sources with spatially inhomogeneous current distribution
- Particle interfaces for coupling of tracking/tracking or tracking/PIC simulations
- ASCII emission data imports based on particle interfaces
- Static-, eigenmode- and multiple external field distributions as additional source fields
- Import of tetrahedral source fields
- Space charge limited, thermionic, fixed and field-induced emission model
- Oblique emission
- Secondary electron emission as material property
- Definable material transparency of sheets for particles
- Consideration of space charge via gun iteration
- Analysis of extracted particle current and space charge
- Monitoring of beam cross-section, phase-space diagram and other statistical data of the beam
- Emittance calculation
- Thermal coupling (export of thermal loss distribution from crashed particles)
- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for remote computations
- Coupled simulations with the Thermal Solver from CST MPHYSICS® STUDIO
- Support of Linux batch modus

Particle-in-Cell Simulator

- Arbitrary shaped particle source surfaces
- Circular particle sources with spatially inhomogeneous current distribution
- Circular particle source in open boundaries
- Gaussian-, DC-, field induced- and explosive emission model
- Oblique emission
- Particle interfaces for coupling of tracking and PIC simulations
- ASCII emission data imports based on particle interfaces
- Static-, eigenmode- and multiple external field distributions as additional source fields
- Import of tetrahedral source fields
- Automatic detection of multipaction breakdown
- Support of GPU acceleration
- Single node parallelization
- Support of Linux batch modus
- Online visualization of intermediate results during simulation
- Calculation of field distributions as a function of time or at multiple selected frequencies from one simulation run
- Time domain monitoring of particle positions and momenta
- Time domain monitoring of output power
- Time domain monitoring of particle current density
- Phase space monitoring
- Emittance calculation
- Secondary electron emission as material property

- Definable material transparency of sheets for particles
 - Isotropic and anisotropic material properties
 - Frequency dependent material properties with arbitrary order for permittivity and permeability as well as a material parameter fitting functionality
 - Field-dependent microwave plasma and gyrotropic materials (magnetized ferrites)
 - Non-linear material models (Kerr, Raman)
 - Surface impedance models (tabulated surface impedance, Ohmic sheet, lossy metal, corrugated wall, material coating)
 - Frequency dependent multilayered thin panel materials (isotropic and symmetric)
 - Time dependent conductive materials
-
- Port mode calculation by a 2D eigenmode solver in the frequency domain
 - Efficient calculation of higher order port modes by specifying target frequency
 - Automatic waveguide port mesh adaptation
 - Multipin ports for TEM mode ports with multiple conductors
 - User defined excitation signals and signal database
-
- Charge absorbing open boundaries for CPU solver
 - High performance radiating/absorbing boundary conditions
 - Conducting wall boundary conditions
 - Inhomogeneous port accuracy enhancement for highly accurate port termination
-
- Calculation of various electromagnetic quantities such as electric fields, magnetic fields, surface currents, power flows, current densities, power loss densities, electric energy densities, magnetic energy densities, voltages or currents in time and frequency domain
 - Calculation of time averaged power loss volume monitors
 - Calculation of time averaged surface losses
-
- Discrete edge and face elements (lumped resistors) as ports
 - Ideal voltage and current sources
 - Discrete edge and face R, L, C, and (nonlinear) diode elements at any location in the structure
-
- Automatic parameter studies using built-in parameter sweep tool
 - Automatic structure optimization for arbitrary goals using built-in optimizer
 - Network distributed computing for remote computations
 - Coupled simulations with the Thermal Solver from CST MPHYSICS® STUDIO

Wakefield Simulator

- Particle beam excitation for ultra-relativistic and non-relativistic beams
 - Transmission line injection scheme (improved dispersion characteristics)
 - Arbitrary particle beam shapes for ultra-relativistic beams
 - Automatic wake-potential calculation
 - Automatic wake-impedance, loss and kick factor calculation
 - Wakefield postprocessor allows to recompute wake impedances
 - Mesh settings for particle beams
 - Direct and two indirect wake-integration methods available
-
- MPI Cluster parallelization via domain decomposition
 - Support of Linux batch modus
-
- Efficient calculation for loss-free and lossy structures

- Calculation of field distributions as a function of time or at multiple selected frequencies from one simulation run
- Adaptive mesh refinement in 3D
- Isotropic and anisotropic material properties
- Frequency dependent material properties
- Gyrotropic materials (magnetized ferrites)
- Surface impedance model for good conductors

- Port mode calculation by a 2D eigenmode solver in the frequency domain
- Automatic waveguide port mesh adaptation
- Multipin ports for TEM mode ports with multiple conductors

- High performance absorbing boundary conditions also for charged particle beams
- Conducting wall boundary conditions

- Calculation of various electromagnetic quantities such as electric fields, magnetic fields, surface currents, power flows, current densities, power loss densities, electric energy densities, magnetic energy densities, voltages or currents in time and frequency domain
- Calculation of time averaged power loss volume monitors
- Calculation of time averaged surface losses
- Discrete edge and face elements (lumped resistors) as ports
- Ideal voltage and current sources
- Discrete edge and face R, L, C, and (nonlinear) diode elements at any location in the structure

- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations, parameter sweeps and multiple port/mode excitations
- Coupled simulations with the Thermal Solver from CST MPHYSICS® STUDIO

Eigenmode Simulator

- Calculation of modal field distributions in closed loss-free or lossy structures
- Support of hexahedral meshes as well as linear and curved tetrahedral meshes
- Isotropic and anisotropic materials
- Multithread parallelization
- Adaptive mesh refinement in 3D using eigenmode frequencies as stop criteria, with True Geometry Adaptation

- Periodic boundary conditions including phase shift
- Calculation of losses and internal / external Q-factors for each mode (directly or using perturbation method)
- Discrete L,C elements at any location in the structure
- Target frequency can be set (calculation within the frequency interval)
- Calculation of all eigenmodes in a given frequency interval
- Sensitivity analysis with respect to materials and geometric deformations defined by face constraints (with tetrahedral mesh)
- Automatic Lorentz force calculation

- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations and parameter sweeps
- Coupled simulations with the Thermal Solver from CST MPHYSICS® STUDIO

Electrostatics Simulator

- Isotropic and (coordinate-dependent) anisotropic material properties
- Sources: potentials, charges on conductors (floating potentials), uniform volume- and surface-charge densities
- Force calculation
- Capacitance calculation
- Electric / magnetic / tangential / normal / open / fixed-potential boundary conditions
- Perfect conducting sheets and wires
- Discrete edge capacitive elements at any location in the structure
- Adaptive mesh refinement in 3D
- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations, parameter sweeps and remote calculations
- Coupled simulations with the Mechanical Solver from CST MPHYSICS® STUDIO

Magnetostatics Simulator

- Isotropic and (coordinate-dependent) anisotropic material properties
- Nonlinear material properties
- Laminated material properties
- Sources: coils, permanent magnets, current paths, external fields, stationary current fields
- Discrete edge inductances at any location in the structure
- Force calculation
- Inductance calculation
- Flux linkages
- Electric / magnetic / tangential / normal / open boundary conditions
- Adaptive mesh refinement in 3D
- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations, parameter sweeps and remote calculations
- Coupled simulations with the Mechanical Solver from CST MPHYSICS® STUDIO

Visualization and Secondary Result Calculation

- Multiple 1D result view support
- Import and visualization of external xy-data
- Copy / Paste of xy-datasets
- Fast access to parametric data by interactive tuning sliders
- Automatic parametric 1D result storage
- Displays port modes (with propagation constant, impedance, etc.)
- Various field visualization options in 2D and 3D for electric fields, magnetic fields, power flows, surface currents, etc.
- Animation of field distributions
- Particle and secondary electrons vs. time 1D plots (PIC)
- Current/Power 1D plot of emitted and absorbed particles (PIC)
- Wave-Particle Power Transfer (PIC)
- Animation of 2D and 3D particle positions / momenta (PIC)

- Visualization of 3D particle trajectories (Tracking)
- Display of source definitions in 3D
- Display of nonlinear material curves in xy-plots
- Display of material distributions for materials with nonlinear permeability
- Animation of field distributions
- Display and integration of 2D and 3D fields along arbitrary curves
- Integration of 3D fields across arbitrary faces
- Hierarchical result templates for automated extraction and visualization of arbitrary results from various simulation runs. These data can also be used for the definition of optimization goals.

Result Export

- Export of result data such as fields, curves, etc. as ASCII files
- Export of particle data as ASCII files
- Export screen shots of result field plots

Automation

- Powerful VBA (Visual Basic for Applications) compatible macro language including editor and macro debugger
- OLE automation for seamless integration into the Windows environment (Microsoft Office®, MATLAB®, AutoCAD®, MathCAD®, Windows Scripting Host, etc.)

About This Manual

This manual is primarily designed to enable a quick start with CST PARTICLE STUDIO. It is not intended to be a complete reference guide to all the available features but will give you an overview of key concepts. Understanding these concepts will allow you to learn how to use the software efficiently with the help of the online documentation.

The main part of the manual is the *Simulation Workflow* (Chapter 2) which will guide you through the most important features of CST PARTICLE STUDIO. We strongly encourage you to study this chapter carefully.

Document Conventions

- Buttons that should be pressed within dialog boxes are always written in italics, e.g. *OK*.
- Key combinations are always joined with a plus (+) sign. *Ctrl+S* means that you should hold down the “Ctrl” key while pressing the “S” key.
- The program’s features can be accessed through a Ribbon command bar at the top of the main window. The commands are organized in a series of tabs within the Ribbon. In this document a command is printed as follows: *Tab name: Group name* \Rightarrow *Button Name* \Rightarrow *Command name*. This means that you should activate the proper tab first and then press the button *Command name*, which belongs to the group *Group name*. If a keyboard shortcut exists it is shown in brackets after the command.
Example: *View: Visibility* \Rightarrow *Wire Frame (Ctrl+W)* 
- The project data is accessible through the navigation tree on the left side of the application’s main window. An item of the navigation tree is referenced in the following way: *NT: Tree folder* \Rightarrow *Sub folder* \Rightarrow *Tree item*.
Example: *NT: 1D Results* \Rightarrow *Port Signals* \Rightarrow *i1*

Your Feedback

We are constantly striving to improve the quality of our software documentation. If you have any comments regarding the documentation, please send them to your local support center. If you do not know how to contact the support center near you, send an email to info@cst.com.

Chapter 2 — Simulation Workflows

CST PARTICLE STUDIO is designed for ease of use. However, to get started quickly, you need to know a few things. The main purpose of this chapter is to provide an overview of the software's capabilities. Read this chapter carefully, as this may be the fastest way to learn how to use the software efficiently.

This chapter covers three different workflow examples for particle tracking, Particle in Cell (PIC) and wakefield computations:

- 1. Workflow Example: Particle Tracking**
 - 1.1. Model and simulate a simple electron gun, including a particle simulation (static approximation)
 - 1.2. Parameter studies of the model and automatic optimization of the structure
 - 1.3. Symmetry conditions
- 2. Workflow Example: Particle in Cell**
 - 2.1. Model and simulate a simple output cavity
- 3. Workflow Example: Wakefield**
 - 3.1. Model and simulate a simple pillbox cavity

Simulation Workflow: Particle Tracking

The following example shows a fairly simple tracking simulation. Studying this example carefully will allow you to become familiar with many standard operations that are necessary to perform a tracking simulation within CST PARTICLE STUDIO.

Go through the following explanations carefully even if you are not planning to use the software for gun simulations. Only a small portion of the example is specific to this particular application type since most of the considerations are general to all solvers and application domains.

At the end of this example, you will find some remarks concerning the differences between the typical simulation procedures for electrostatic and magnetostatic calculations and some useful hints for setting up the particle tracking and gun algorithm.

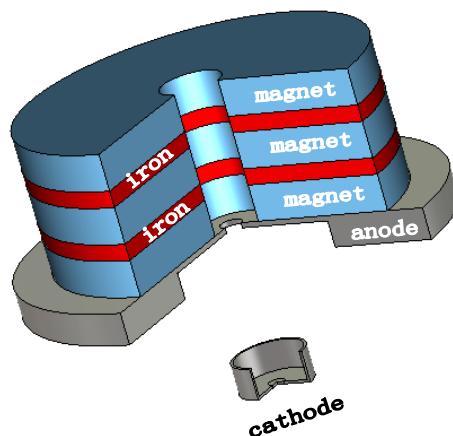
The following explanations always describe the menu-based way to open a particular dialog box or to launch a command. Whenever available, the corresponding toolbar item is displayed next to the command description. Due to the limited space in this manual, the shortest way to activate a particular command (i.e. by pressing a shortcut key or activating the command from the context menu) is omitted. You should regularly open the context menu to check available commands for the currently active mode.

The Structure

Usually an electron gun is only one part of a complex device, for example a particle accelerator. The gun is used to create a collimated particle beam, so that other parts of the device have a beam of good quality.

The way this gun works is quite simple. Electrons are emitted from a cathode by a particle source based on space charge limited emission. These particles are accelerated and focused by an anode. Additional focusing is realized by a set of magnets behind the anode.

The following picture shows the structure of interest. It has been sliced open to aid visualization. Anode and cathode consist of perfect electrical conductor (PEC) material whereas the magnetic structure above the anode consists of iron and permanent magnets.



Before you start modeling the structure, let's spend a few moments discussing how to describe this structure efficiently.

At first, CST PARTICLE STUDIO allows you to define the properties of the background material. Anything you do not fill with a particular material will automatically be filled with the background material. For this structure it is sufficient to model anode, cathode, two iron discs and three permanent magnets of the electron gun. The background properties will be set to vacuum.

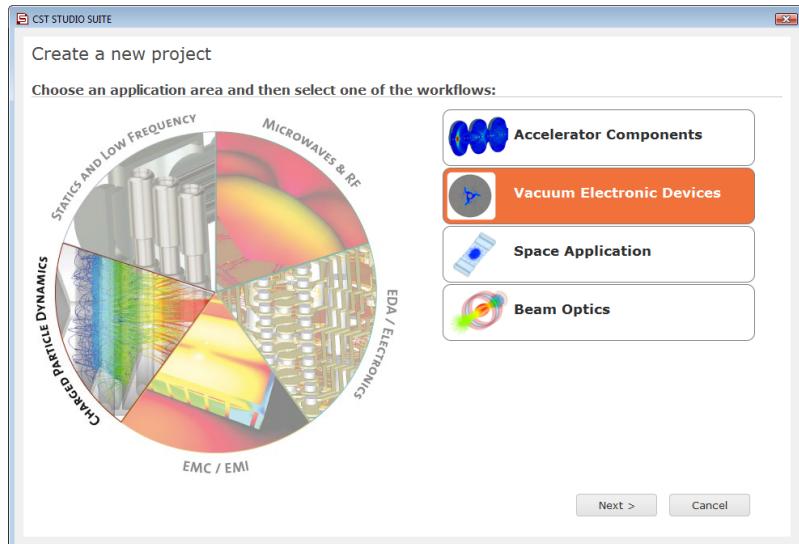
Your method of describing the structure should therefore be as follows:

1. Model cathode and anode of the electron gun.
2. Model the two iron discs.
3. Model the three permanent magnets.

Create a New Project

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

Next you should choose the application area, which is *Charged Particle Dynamics* for the example in this tutorial and then select the workflow by double-clicking on the corresponding entry.

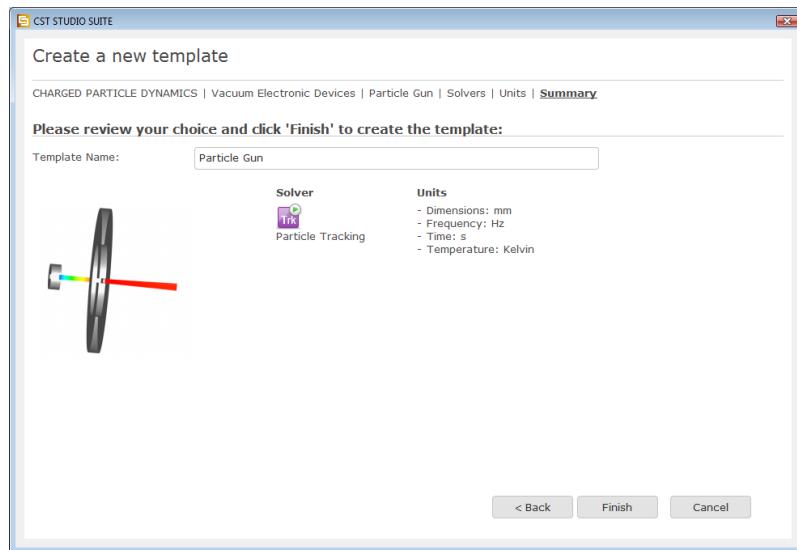


For the electron gun, please select *Vacuum Electronic Devices* \Rightarrow *Particle Gun* \Rightarrow *Particle Tracking* .

At last you are requested to select the units which fit your application best. For this example, please select the dimensions as follows:

Dimensions:	mm
Frequency:	Hz
Time:	s

For the specific application in this tutorial the other settings can be left unchanged. After clicking the *Next* button, you can give the project template a name and review a summary of your initial settings:



Finally click the *Finish* button to save the project template and to create a new project with appropriate settings. CST PARTICLE STUDIO will be launched automatically due to the choice of this specific project template within the application area *Charged Particle Dynamics*.

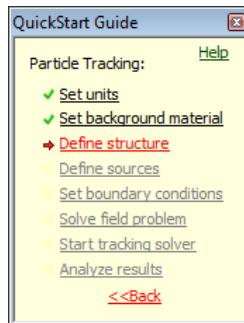
Please note: When you click again on the *File: New and Recent* you will see that the recently defined template appears below the *Project Templates* section. For further projects in the same application area you can simply click on this template entry to launch CST PARTICLE STUDIO with useful basic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

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

Open the Tracking QuickStart Guide

An interesting feature of the online help system is the QuickStart Guide, an electronic assistant that will guide you through your simulation. If it does not show up automatically, you can open this assistant by selecting *QuickStart Guide* from the Help button  in the upper right corner.

The following dialog box should then be visible at the upper right corner of the main view:



As the project template has already set the solver type, units and background material, the *Particle Tracking Analysis* is preselected and some entries are marked as done. The red arrow always indicates the next step necessary for your problem definition. You do not have to follow the steps in this order, but we recommend you follow this guide at the beginning to ensure that all necessary steps have been completed.

Look at the dialog box as you follow the various steps in this example. You may close the assistant at any time. Even if you re-open the window later, it will always indicate the next required step.

If you are unsure of how to access a certain operation, click on the corresponding line. The *QuickStart Guide* will then either run an animation showing the location of the related menu entry or open the corresponding help page.

Define the Units

The *Particle Gun* template has already applied some settings for you. The defaults for this structure type are geometrical units in mm and time in s. You can change these settings by entering the desired settings in the units dialog box (*Home: Settings \Rightarrow Units* ), but for this example you should just leave the settings as specified by the template. Additionally, the used units are also displayed in the status bar:



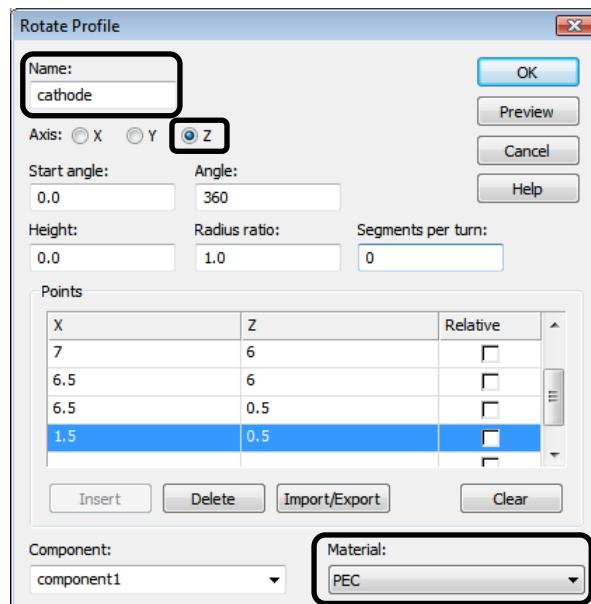
Define the Background Material

As discussed above, the structure will be described within vacuum. The material type *Normal* is set as default background material in the *Particle Gun* template. For this example, you do not need to make any changes as the default properties of the material type *Normal* are those of vacuum. For cases where you do need to change the properties, you may do so in the corresponding dialog box *Simulation: Settings \Rightarrow Background* .

Model the Structure

The basic settings have been made, now we are able to set up the structure. Since the electron gun is rotationally symmetric, a special but very efficient technique can be used to design the structure. First of all the cathode is created.

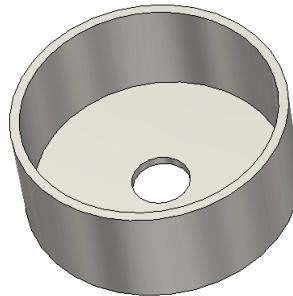
1. Open the *Rotate Profile* dialog box *Modeling: Shapes \Rightarrow  (dropdown list) \Rightarrow Rotate*  to create the cathode.
2. Press the *ESC* button to show the dialog box. Do not click a point in the working plane.



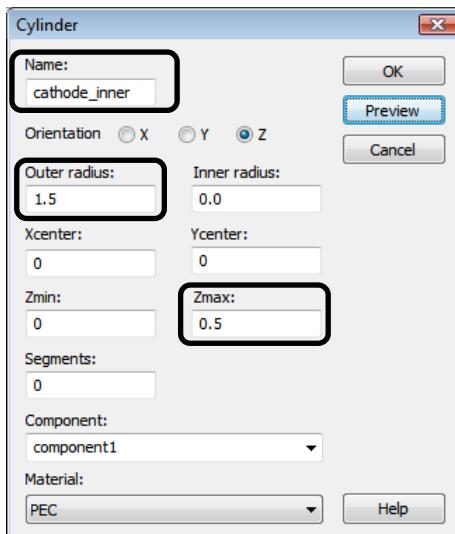
3. Enter the name "cathode" and choose z as axis of rotation. Set the material to PEC. Now enter the polygon data as shown in the table below.

x	z
1.5	0.0
7.0	0.0
7.0	6.0
6.5	6.0
6.5	0.5
1.5	0.5

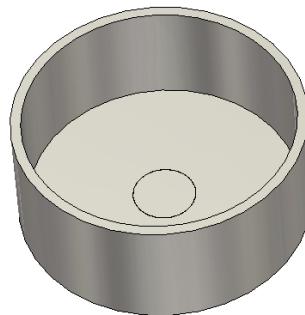
4. You may click the preview button during the construction to get a preview of the solid. This makes it easy to detect any possible mistakes when entering the data. The dialog box should now look like in the picture above. Click the *OK* button to confirm your settings and to construct the cathode.
 5. The structure is displayed in the working plane and now your cathode should look like this:



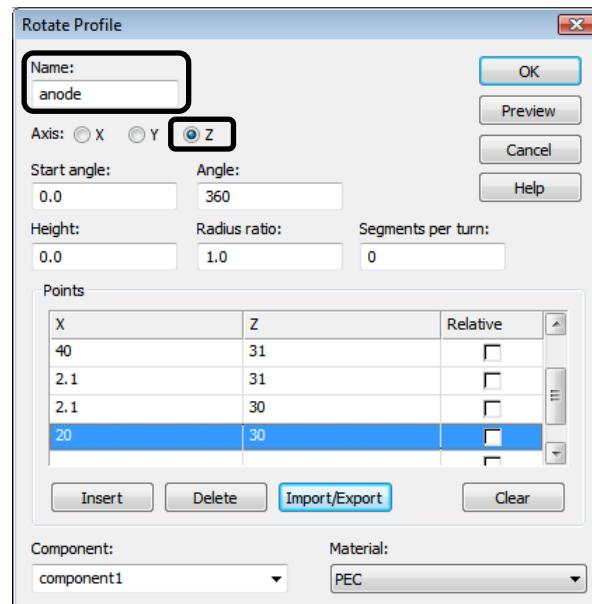
One part of the cathode is still missing, the inner cylinder. We will need this inner cylinder to define the particle source. To create this cylinder, open the Cylinder dialog box *Modeling: Shapes* \Rightarrow *Cylinder*. Press the *ESC* button to show the dialog box.



Change the name the name to "cathode_inner", enter an *Outer radius* of 1.5 and *Zmax* of 0.5. Click the *OK* button to confirm your changes. The cylinder should fit perfectly into the hole of the solid cathode:



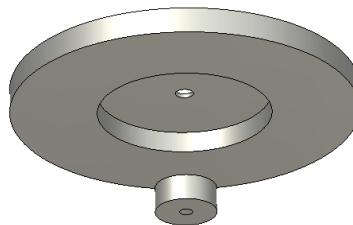
6. The construction is completely finished and now we will construct the anode in the same way as we constructed the outer cathode. Open the *Rotate Profile* dialog box *Modeling: Shapes* \Rightarrow *Rotate*.
7. Press the *ESC* button to show the dialog box. Do not click a point in the working plane.
8. Enter the name "anode" and choose z as axis of rotation. The material PEC should be automatically selected.



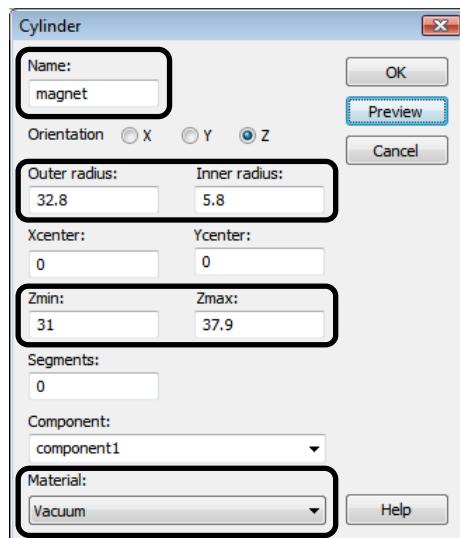
Your dialog box should now look like in the picture above. Now enter the polygon data as shown in the following table:

x	z
20.0	25.0
40.0	25.0
40.0	31.0
2.1	31.0
2.1	30.0
20.0	30.0

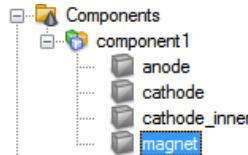
Click the *OK* button to confirm your changes. The creation of the anode is complete and the whole structure should look like this:



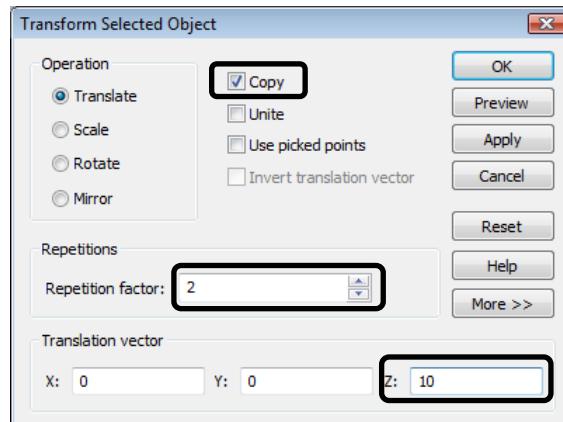
9. As you might have noticed, the magnetic part of the structure is still missing. First, we will construct the three vacuum discs that will serve as permanent magnets. To create one disc, open the Cylinder dialog box *Modeling: Shapes* \Rightarrow *Cylinder*. Press the *ESC* button to show the dialog box.
10. Enter the name "magnet", outer radius 32.8 and the inner radius 5.8. The z range extends from 31 to 37.9 mm. Change the material to vacuum. Click the *OK* button to confirm your changes.



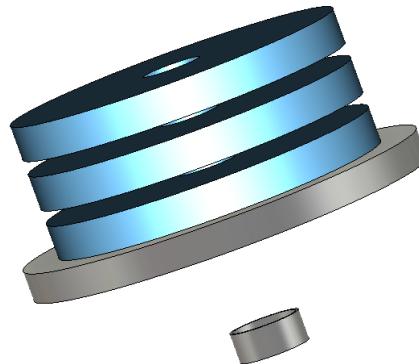
11. Since the same cylinder exists three times, we will use the transform dialog box to create the missing two cylinders. First select the solid "magnet" in the navigation tree *NT: Components* \Leftrightarrow *component 1* \Leftrightarrow *magnet*.



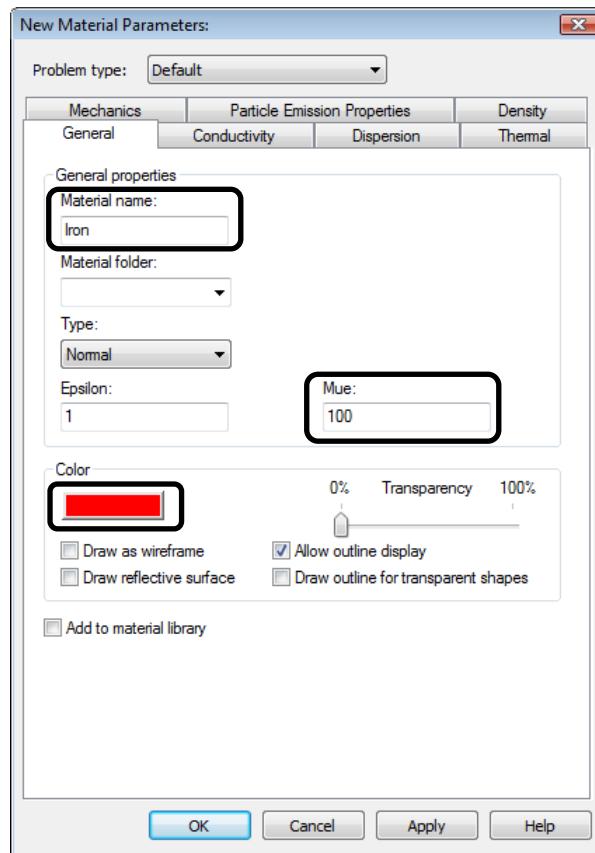
12. Open the *Transform Selected Object* dialog box *Modeling: Tools* \Leftrightarrow *Transform* to copy the cylinder.



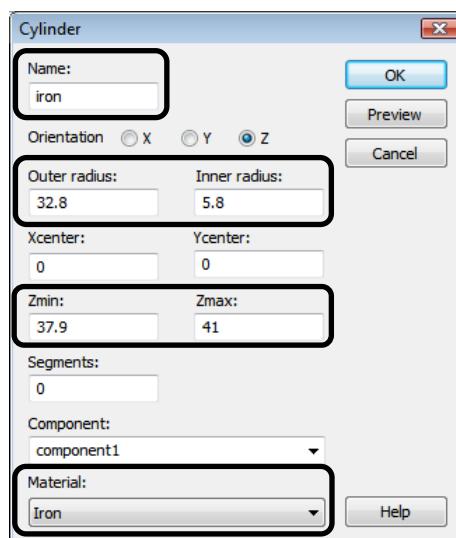
Enable the checkbox *Copy*. Then enter a translation of 10 in z direction. Change the Repetition factor to 2 and click the *OK* button to confirm your changes. The structure should now look like this:



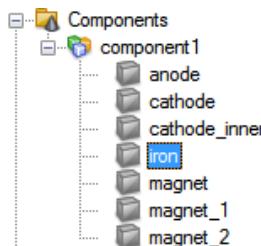
13. Before the iron discs will be defined, we create a new and simple iron material. To do this, open the material dialog box *Modeling: Materials* \Leftrightarrow *New/Edit* \Leftrightarrow *New Material*. Change the *Material name* to "Iron", the *Color* to red and value of *Mue* to 100 like in the picture below. Now we have quickly defined a simple iron material. Click the *OK* button to confirm your changes and to leave this dialog box.



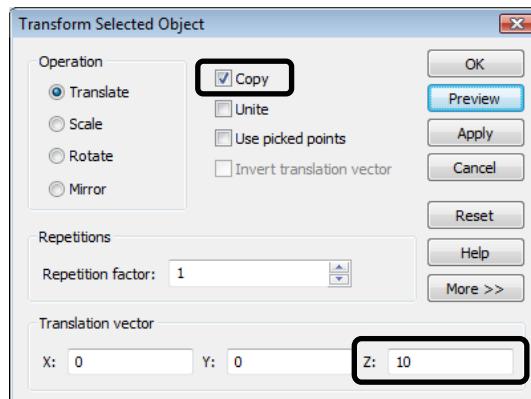
14. The iron discs are created in the same way as the magnets. Open the cylinder dialog box *Modeling: Shapes* \Rightarrow *Cylinder*. Press the *ESC* button to show the dialog box.



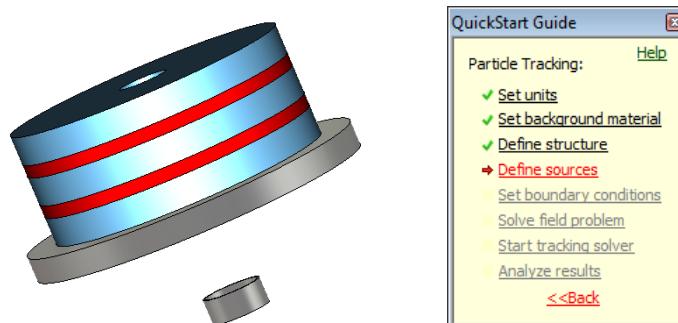
15. Enter the *Name* "iron", an *Outer radius* of 32.8 and *Inner radius* of 5.8. The z range extends from 37.9 to 41 mm. Change the material to the new material "iron". Your dialog box should now look like the picture above.
16. Finally click the *OK* button and confirm your changes. To create the second iron disc we will use the transform mechanism again. Select the solid "iron" in the navigation tree.



17. Open the *Transform Selected Object* dialog box *Modeling: Tools* \Rightarrow *Transform* to copy the cylinder.



18. Select copy and enter a translation of 10 in z-direction. Click the *OK* button to confirm your changes. Now the structure should look like this:



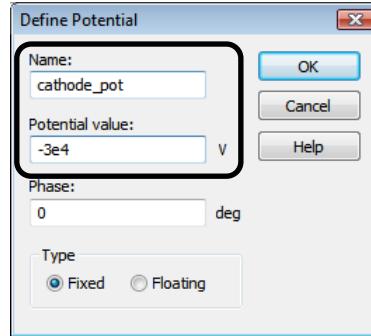
19. The structure creation part is finished and we can start to define the sources, i.e. potentials, magnets and particle sources.

Congratulations! You have just created your first tracking structure within CST PARTICLE STUDIO.

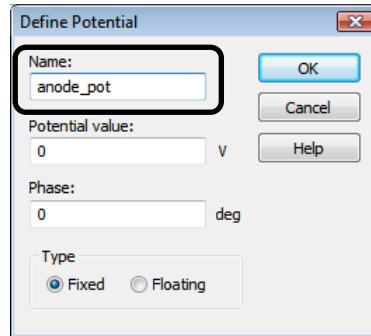
Define Potentials and Magnets

With all components for the electrostatic part of the configuration defined, the appropriate potentials can be set. First define the potentials of the cathode and anode:

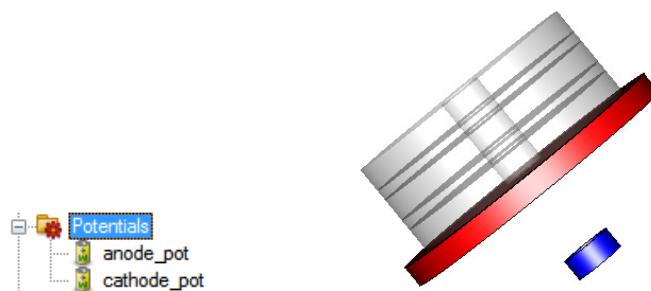
1. Select *Simulation: Sources and Loads* \Rightarrow *Static Sources* \Rightarrow *Electric Potential*  and double-click on the cylindrical surface of the cathode in the working plane. Press the *Return* key to finish your selection and to open the *Define Potential* dialog box.



2. Enter the name "cathode_pot" and a value of -3e4 V. As usual click the *OK* button to confirm your changes.
3. In the same way the potential for the anode is defined. Select *Simulation: Sources and Loads* \Rightarrow *Static Sources* \Rightarrow *Electric Potential*  and double-click on the cylindrical surface of the anode. Press the *Return* key to finish your selection and to open the *Define Potential* dialog box.
4. Enter the name "anode_pot" and a value of 0 V. Click the *OK* button to confirm your changes.

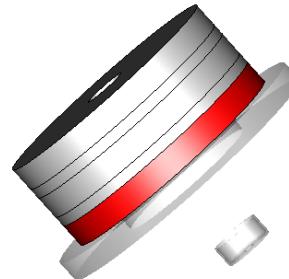


5. If you now select the potential folder in the navigation tree your structure should look like the picture below:

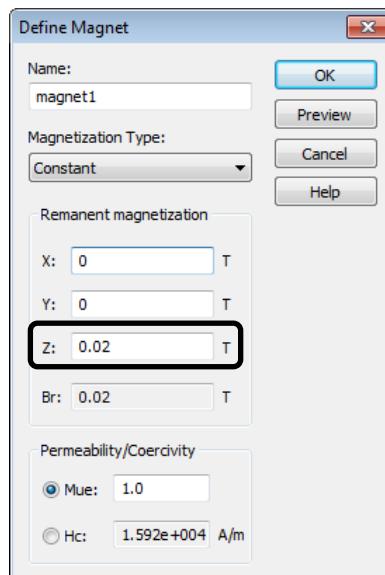


Note: As the solids "cathode" and "cathode_inner" are in direct contact, both solids have always the same potential. That means "cathode_inner" has also a potential of -30 kV.

6. As the potential definition is finished, we will create three permanent magnets for the three vacuum discs. To define the first magnet select *Simulation: Sources and Loads* \Rightarrow *Static Sources* \Rightarrow *Permanent Magnet*
7. Then select the solid that should become a permanent magnet. Thus click double on the vacuum disc named "magnet".



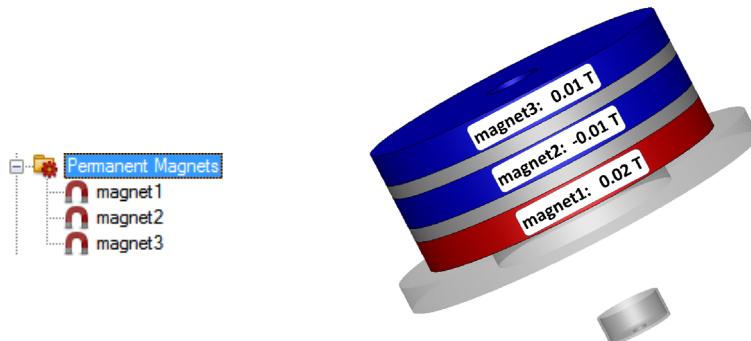
8. The *Permanent Magnet* dialog box opens. Enter a value of 0.02 T in z-direction. Leave other settings unchanged and click *OK* to confirm.



9. In the same way define magnets for the vacuum solids "magnet_1" and "magnet_2" in z-direction. The solid "magnet_1" should be the vacuum disc in the middle of the three discs.

solid	T
magnet	0.02
magnet_1	-0.01
magnet_2	0.01

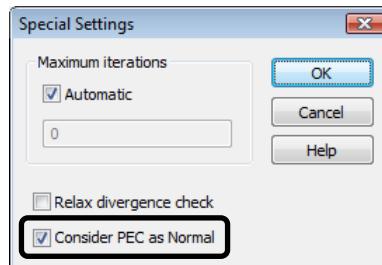
10. If you now select the Permanent Magnets folder in the navigation tree you should see the following picture:



11. Potential and magnet definitions are finished.

In practice it is advisable to visualize and refine the mesh before the particle source is defined. The reason is that the number of emission points of the particle source can depend on the mesh settings. This matter is discussed in detail in the later chapter *Define Particle Sources*.

Note: The option *Consider PEC as Normal* is default only when a Particle Tracking or PIC project template is used - otherwise this checkbox is disabled by default. If you want to change or check this setting, open the *Special Settings* dialog box of the *Magnetostatic Solver Parameters* via *Home: Simulation* \Rightarrow *Setup Solver* \Rightarrow *M-Static Solver* \Rightarrow *Specials*.



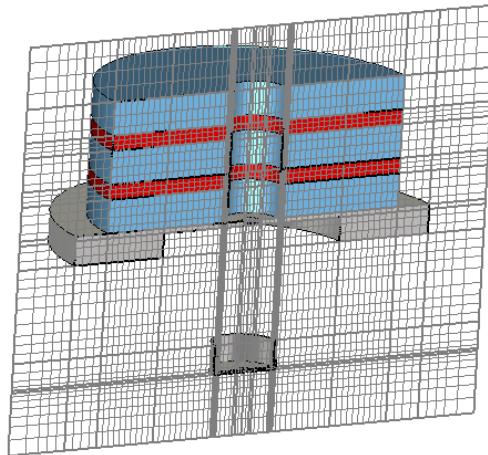
If the checkbox *Consider PEC as Normal* is enabled, PEC materials are considered like normal materials with a permeability μ which can be defined in the material properties of the PEC material.

Visualize and Refine the Mesh

The mesh generation for the structure's analysis is performed automatically based on an expert system. However, in some situations it may be helpful to inspect the mesh to improve the simulation speed by changing the parameters for the mesh generation.

The mesh can be visualized by entering the mesh view *Home: Mesh* \Rightarrow *Mesh View* .

For this structure, the mesh information will be displayed as follows:

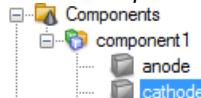


One 2D mesh plane is visible at a time. Because of the symmetry setting, the mesh plane extends across only one half of the structure. You can modify the orientation of the mesh plane by adjusting the selection in the *Mesh: Sectional View* \Rightarrow *Normal* dropdown list or just by pressing the X/Y/Z keys. Move the plane along its normal direction using the Up/Down cursor keys. The current position of the plane will be shown in the *Mesh: Sectional View* \Rightarrow *Position* field.

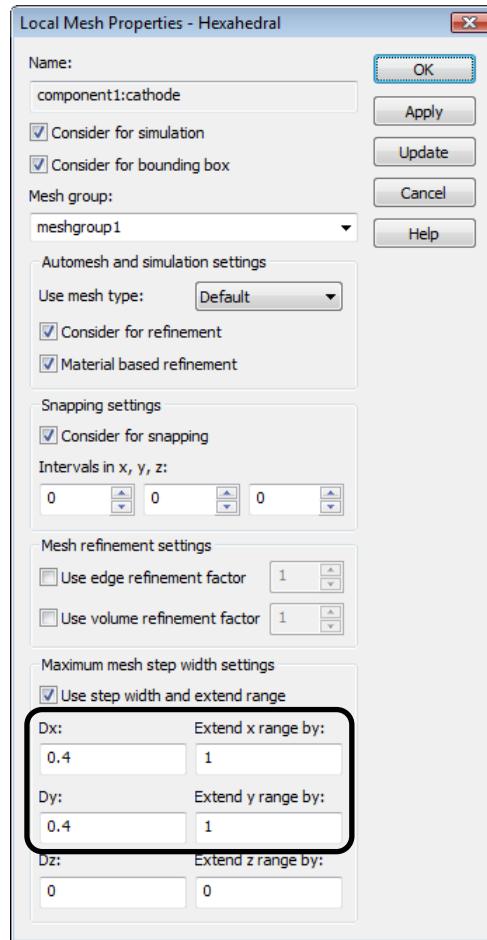
There are some thick mesh lines shown in the mesh view. These mesh lines represent important planes (so-called snapping planes) at which the expert system finds it necessary to place mesh lines. You can control these snapping planes in the Special Mesh Properties dialog by selecting *Simulation: Mesh* \Rightarrow *Global Properties*  \Rightarrow *Specials* \Rightarrow *Snapping*.

In a lot of cases the automatic mesh generation will produce a reasonable initial mesh, but in our case we will refine the mesh in the cathode region to have a finer mesh resolution for the particle beam.

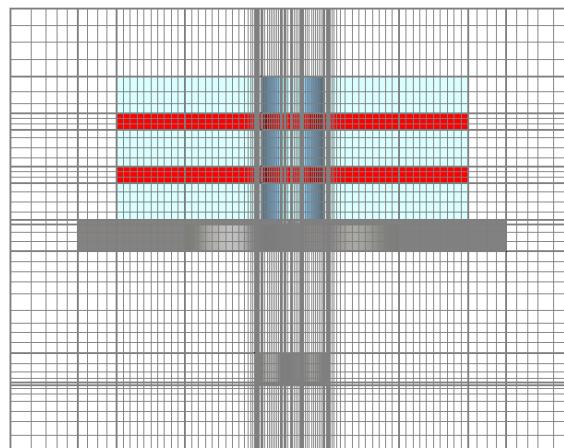
1. Make sure you are in the mesh view mode. Select the solid *cathode* in the navigation tree *NT: Components* \Rightarrow *component1* \Rightarrow *cathode*.



2. Open the dialog box *Mesh: Mesh Control* \Rightarrow *Local Properties* to modify the local mesh settings of the cathode. Change *Dx* and *Dy* to a value of 0.4. Extend the x and y range by a number of 1.



3. Confirm your changes as usual by clicking on the OK button. The dialog box closes and you can see the modified mesh.



The number of mesh cells should be 497,536. You can get this information from the status bar.

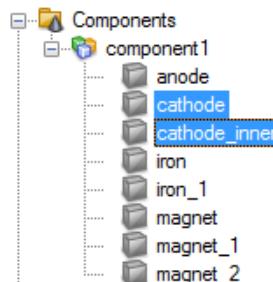


You should now leave the mesh inspection mode by toggling *Mesh: Close* \Leftrightarrow *Close Mesh View* .

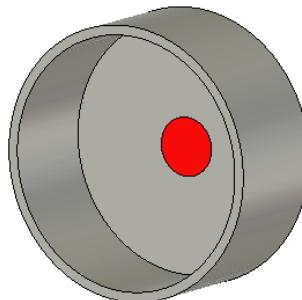
Define Particle Sources

A particle source is a shaped surface of a component where charged particles enter the computational domain under a specific emission condition, which is determined by the emission model settings. Such a source is often located on the surface of a PEC solid, but it can also be defined on the surface of any arbitrary material. In our case the particle source will be placed on the inner cathode. To facilitate the selection of the surface of the inner cathode, some solids will be hidden.

1. Select "cathode" and "cathode_inner" in the navigation tree. Use the *Shift* key for multi-selection. Select the option *View: Visibility* \Leftrightarrow *Hide* \Leftrightarrow *Hide Unselected*. Now we are able to define the particle source.

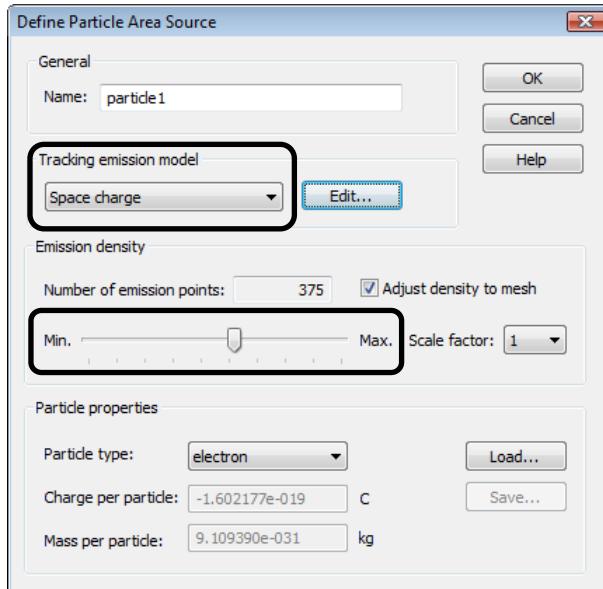


2. Select *Simulation: Sources and Loads* \Leftrightarrow *Particle Sources* \Leftrightarrow *Particle Area Source* and select the inner surface of the solid "cathode_inner" by double-clicking on it. Make sure that the surface is highlighted when you move the mouse cursor away from the surface.



3. After selecting the emission surface, press the *Return* key to open the particle source dialog box. In this dialog box, the particle type and particle density at the previously selected surface are adjustable. The blue points in the preview

visualize the particle emission points. Move the slider *Particle density* to the middle position and change the *Tracking emission model* to *Space charge*. An increase of the number of emission points leads in this case to smoother current density. The checkbox *Adjust density to mesh* should be enabled if the emission model *space charge* is chosen. Otherwise the number of emission points might be too low to obtain good simulation results.

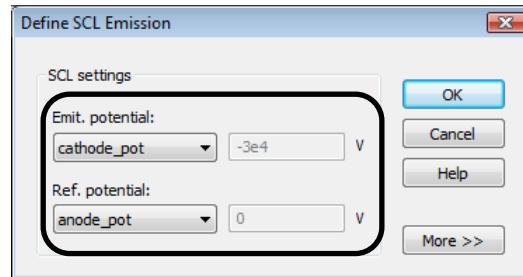


The particle density is adjustable with the help of the particle density slider. The option *Adjust density to mesh* should usually be activated if tracking calculations are performed. This setting guarantees an appropriate sampling rate with respect to the mesh and therefore directly affects the accuracy of the simulation.

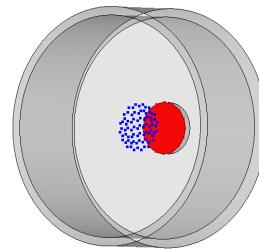
Note:

As seen in the lower part of the dialog box, standard or user-defined particle types can be specified. A particle definition library allows you to export such user-defined particle definitions to a database and also to import them. This library is accessible through the *Load* and *Save* buttons. In this example, we keep the default particle type *electron*.

4. Move the *Number of emission points* slider until the number is 375 or close to 375. To change the space charge emission model settings click the *Edit* button close to the emission model drop down list. The *Define SCL Emission* dialog box opens:



5. An emission model describes the conditions particles need to fulfill in order to be emitted into free space. For instance, the space charge emission model allows particles to be emitted as long as an electric field perpendicular to the emitting surface is present. If not already preselected, make the following adjustments inside the dialog box: Change the emitting potential to "cathode_pot" and the reference potential to "anode_pot". Click the *OK* button to confirm your changes. The particle source should now look like this:



Note:

The red triangular mesh shows the discretization of the cathode surface, while the blue points visualize the particles' start positions for the simulation. In this case the emission model *Space charge limited* requires the start positions to be shifted a little bit away from the cathode surface. This shifting is done automatically depending on the mesh close to the cathode.

6. We finished the particle source definition and leave the *Define Particle Source on Area* dialog box by clicking the *OK* button again.
7. Since some solids are currently hidden, we have to unhide them to see the whole structure again. Select *View: Visibility* \Rightarrow *Show* \Rightarrow *Show All*. It is often helpful to hide some solids in order to select faces inside a structure.

The particle source is now defined and ready for emission. Before you continue, have a look at the QuickStart Guide to see the next steps.

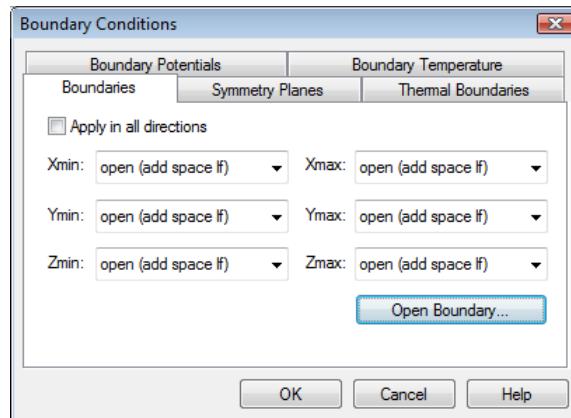


The point “*Set boundary conditions*” already has been set to the status finished as the boundaries were set by the *Particle Gun* template. Nevertheless, the boundary conditions will be discussed in the next section to illustrate the basics of the boundary condition setup.

Define Boundary Conditions

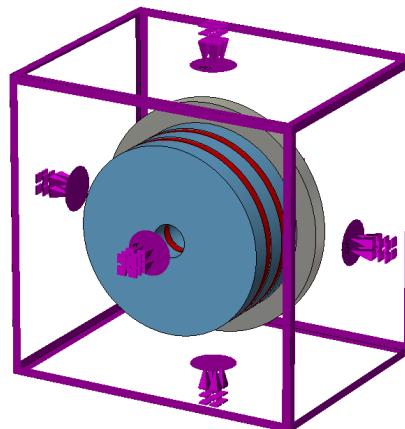
The simulation will be performed only within the bounding box of the structure, the so-called computational domain. You can specify certain boundary conditions for each plane (X_{\min} , X_{\max} , Y_{\min} , Y_{\max} , Z_{\min} , Z_{\max}) of the computational domain. These boundary conditions reflect the appropriate behavior of the surrounding world.

The boundary conditions are specified in a dialog box which can be opened by choosing *Simulation: Settings \Rightarrow Boundaries*.



While the boundary dialog box is open, the boundary conditions are visualized in the structure view as shown in the next picture.

You can change boundary conditions from within the dialog box or interactively in the view. Select a boundary by double-clicking on the boundary symbol within the view and select the appropriate type from the context menu.

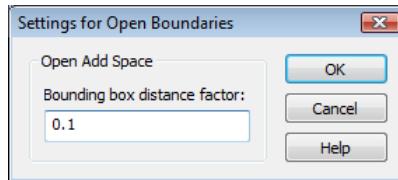


The following table gives an overview of available boundary conditions and their effect on the tangential and normal component of the electric and magnetic fields:

Boundary type	Electric field component		Magnetic field component	
	tang. comp.	norm. comp.	tang. comp.	norm. comp.
electric	0	exists	exists	0
magnetic	exists	0	0	exists
tangential	exists	0	exists	0
normal	0	exists	0	exists
open	exists	exists	exists	exists

In our case we want to use open boundaries in all directions. As we use the *Charged Particle Dynamics* template, the default boundaries are already set to open.

Furthermore, some extra space is added between the structure and the open boundaries. Click the button *Open Boundary* to check this setting.

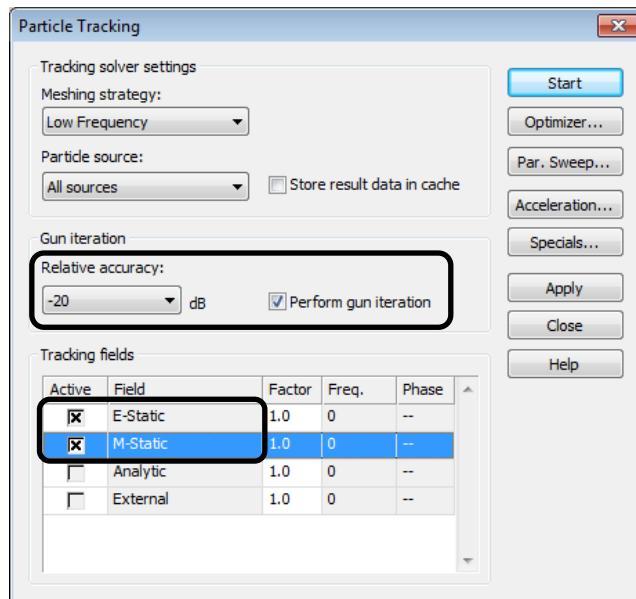


The distance of this extra space is the length of the bounding box diagonal times the user defined factor, in our case 0.1. This value is also defined in the *Charged Particle Dynamics* template. Click *Cancel* to leave this setting unchanged. Click *Cancel* again to leave the *Boundary Conditions* dialog box.

Note: There are two ways to create some space (background material) between structure and boundaries. The first way is described above. Alternatively, some extra space can be defined in the *Background Properties* dialog box. You can have a look in the paragraph *Define the Background Material*.

Start the Simulation

After having defined all necessary parameters, you are ready to perform your first simulation. The simulation is started from within the particle tracking solver control dialog box: *Simulation: Solver* \Rightarrow *Setup Solver* .



In this dialog box you can specify the settings of the particle tracking solver and start the simulation process. If multiple particle sources are defined, you can choose between a simulation where all sources emit particles and a simulation where only a single source is activated. Enable the *Perform gun iteration* option so that the iterative gun solver algorithm is activated and set the *relative accuracy* to be -20 dB. Thus the tracking solver does not just track the particles once through the computational domain. Instead, the solver iteratively repeats an electrostatic calculation and then tracks the particles until the desired accuracy of the space charge deviation between two successive iterations is reached.

The *Tracking fields* box lists all electromagnetic fields that are available for the particle-tracking solver. To consider a field for the tracking process check the *Active* checkbox. The *Factor* setting determines the relative strength of the field values for the particle force calculation. Thus, these factors offer a quick and simple method to analyze what happens if the field strength changes. Activate the electrostatic and the magnetostatic field and set the corresponding factors to 1.

Now you can start the simulation procedure by clicking the *Start* button in the particle tracking dialog box. A few progress bars will appear to keep you up to date with the solver's progress.

As you can see in the next paragraph, the complete solving procedure consists of three to four parts, depending upon the selected post processing steps. Part two (electrostatic solver) and part three (particle tracking) are repeated iteratively until the *relative accuracy* condition specified in the *gun iteration* section is reached.

1. Magnetostatic Solver

- 1.1. *Checking model*: During this step, your input model is checked for errors such as invalid overlapping materials, etc.
- 1.2. *Calculating matrix and dual matrix*: During these steps, the system of equations, which will subsequently be solved, is set up.
- 1.3. *M-Static solver is running*: During this stage a linear equation solver calculates the field distribution inside the structure.

2. Electrostatic Solver

- 2.1. *Calculating matrix and dual matrix*: During these steps the system of equations, which will subsequently be solved, are set up.
- 2.2. *E-Static solver is running*: During this stage a linear equation solver calculates the field distribution inside the structure.

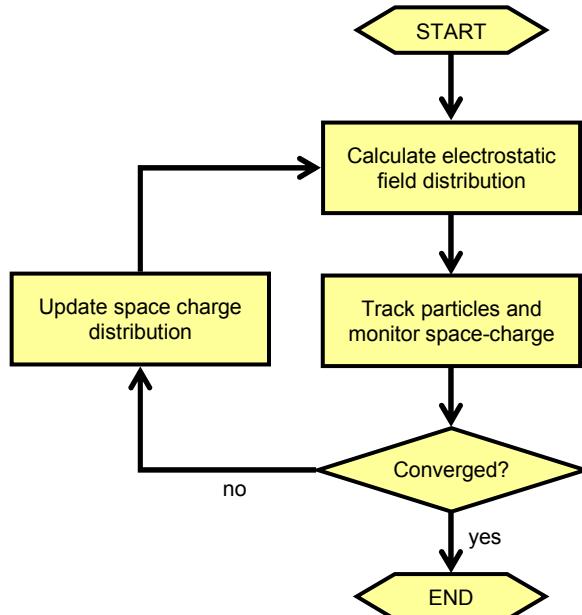
3. Particle Tracking

- 3.1. *Initializing tracking solver*: The data structure for the collision detection of particles with solids is constructed.
- 3.2. *Tracking Solver is running*: The particles are emitted and tracked through the computational domain.

4. Post Processing

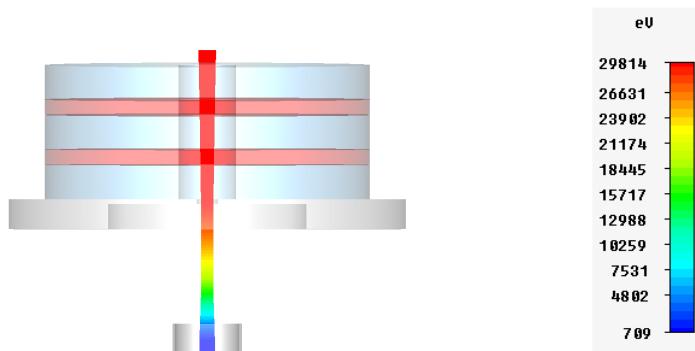
- 4.1. From the field distribution, additional results like the inductance matrix or the energy within the computational domain can be calculated.

After a few repetitions of steps two and three, the desired accuracy of -20 dB of the gun iteration is reached, i.e. the relative difference of the space charge distribution between two consecutive solver runs is less than -20 dB. The algorithm of the iterative gun solver and its convergence condition are explained by the following diagram:

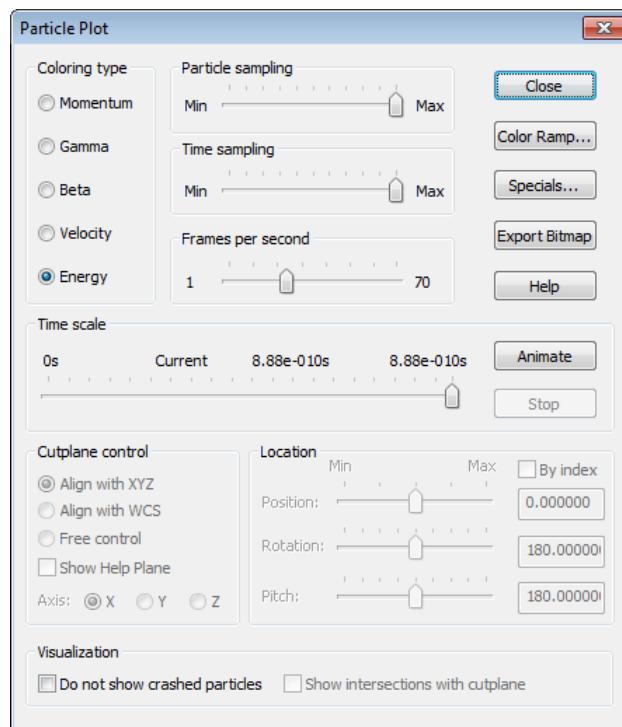


Analyze the Results

In tracking applications users are often interested in the particle beam behavior. To have an overview of the particle movement a 3D visualization of the trajectories is available in the navigation tree *NT: 2D/3D Results* \Rightarrow *Trajectory* \Rightarrow *particle1*. The trajectories should now look like in the following picture:



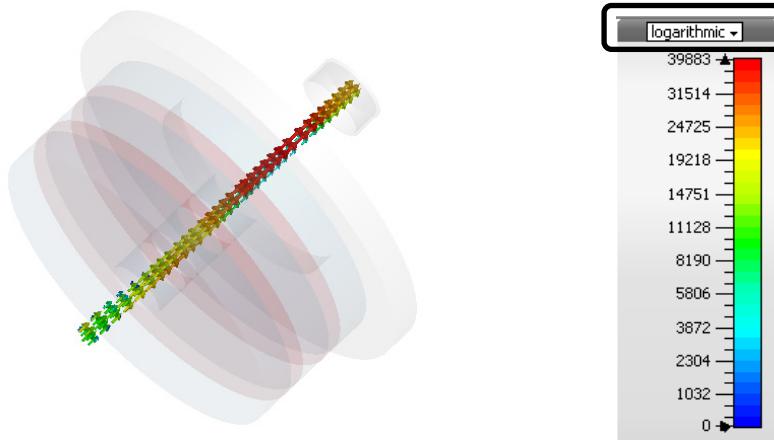
Colors indicate the energy of the particles. The color ramp is needed to assign values to the colors you see in the plot. There are lots of options to modify this plot using the *Particle Plot* properties dialog box *2D/3D Plot: Plot Properties* \Rightarrow *Properties*



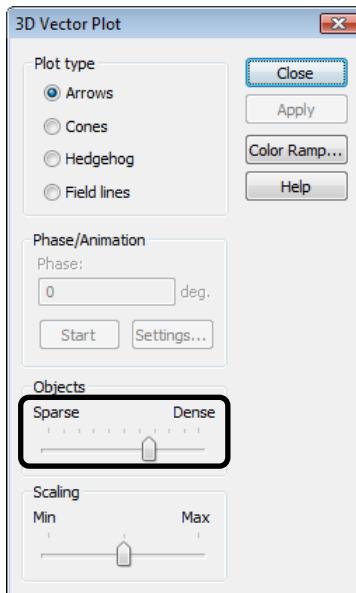
Open the dialog box and change some settings, for example the coloring type. Click the *Animate* button to see the movement of the particles. Detailed explanations can be

obtained from the online help. Click the *Help* button to open the online help in your browser. If you like to close this dialog box, click the *Close* button.

Field plots are also available in the navigation tree. To obtain the current density of the particle beam select *NT: 2D/3D Results* \Rightarrow *Particle Current Density* in the navigation tree. Hover over the color ramp and change the scale to logarithmic.

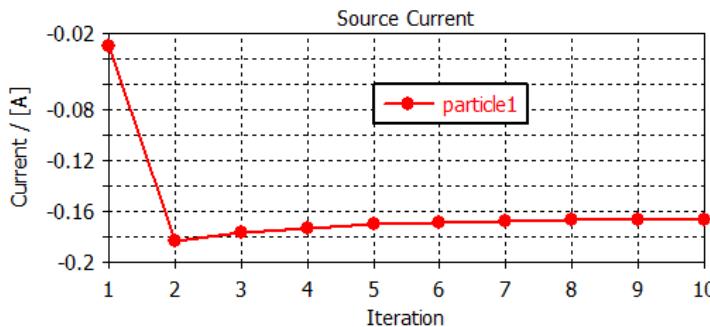


Further plot settings can be changed in the *3D Vector Plot* dialog box. This can be opened as usual via *2D/3D Plot: Plot Properties*



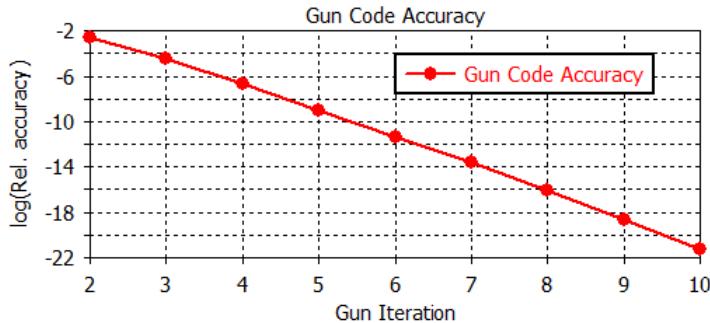
To create the field plot above the density slider has to be shifted to the right. Try to change some settings. Click the *Close* button to leave this dialog box.

In the case of gun simulations with space charge limited emission the emitted current is an important parameter. The 1D result plot emitted current versus gun iteration *NT: 1D Results* \Rightarrow *Particle Source Current* \Rightarrow *particle1* is available in the navigation tree:

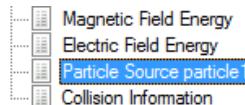


This 1D result offers you the possibility to control the emission process. Thus it is very helpful that this plot is already available during the gun iteration.

Another plot is also available during the gun iteration, the gun code accuracy. If the user defined accuracy is reached, the iterative gun solver stops. To get this 1D result plot select the folder *NT: 1D Results* \Rightarrow *Convergence* \Rightarrow *Tracking Solver* in the navigation tree:



Apart from this 1D graph, the development of the current is listed in a text file, which is accessible in the solver note *NT: Particle Source particle1*.



The text file contains the emitted current, permeance and charge for each iteration step:

```
Result: Particle Source particle1

1. Iter.: Current = -2.896e-002 A, Perveance = -5.573e-009 A/V^(3/2), Charge = -4.830e-015 C
2. Iter.: Current = -1.828e-001 A, Perveance = -3.518e-008 A/V^(3/2), Charge = -3.049e-014 C
3. Iter.: Current = -1.763e-001 A, Perveance = -3.392e-008 A/V^(3/2), Charge = -2.940e-014 C
4. Iter.: Current = -1.722e-001 A, Perveance = -3.313e-008 A/V^(3/2), Charge = -2.871e-014 C
5. Iter.: Current = -1.696e-001 A, Perveance = -3.263e-008 A/V^(3/2), Charge = -2.828e-014 C
6. Iter.: Current = -1.679e-001 A, Perveance = -3.231e-008 A/V^(3/2), Charge = -2.800e-014 C
7. Iter.: Current = -1.669e-001 A, Perveance = -3.211e-008 A/V^(3/2), Charge = -2.783e-014 C
8. Iter.: Current = -1.662e-001 A, Perveance = -3.198e-008 A/V^(3/2), Charge = -2.772e-014 C
9. Iter.: Current = -1.658e-001 A, Perveance = -3.190e-008 A/V^(3/2), Charge = -2.765e-014 C
10. Iter.: Current = -1.655e-001 A, Perveance = -3.185e-008 A/V^(3/2), Charge = -2.760e-014 C
```

The collision information *NT: Collision Information* could be also very interesting for you, because this file contains e.g. information about the power that is absorbed by a solid.

## Collision information of particle source particle1			
Component	Solid	Current (A)	Power (W)
component1	anode	0.000e+000	0.000e+000
component1	cathode	0.000e+000	0.000e+000
component1	cathode_inner	0.000e+000	0.000e+000
component1	iron	0.000e+000	0.000e+000
component1	iron_1	0.000e+000	0.000e+000
[Background]	Boundary	1.655e-001	4.737e+003

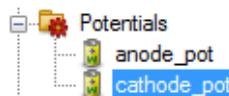
In this case the background consists of vacuum, thus all particles are absorbed by the boundary of our calculation domain.

Parameterization of the Model

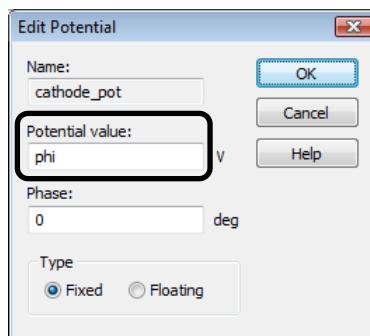
The previous steps demonstrated how to enter and analyze a simple structure. However, structures are usually analyzed to improve their performance. This procedure may be called "design" in contrast to the "analysis" done before.

After you get some information on how to improve the structure, you will learn how to optimize the structure's parameters. This could be done by modifying each parameter manually, but this of course is not the best solution. CST PARTICLE STUDIO offers various options to describe the structure parametrically in order to change the parameters easily.

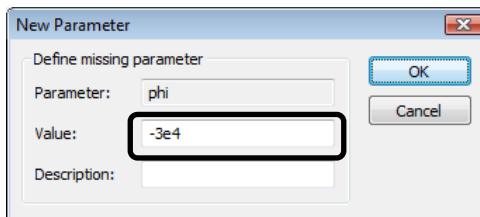
Let us assume you are interested in the dependency of the emitted current on the cathode's potential. To obtain this dependency, first of all the potential has to be parameterized. Thus double click on the potential *NT: Potential* \Leftrightarrow *cathode_pot* in the navigation tree.



The *Edit Potential* dialog box opens and the potential can be edited. Instead of a number type the string "phi" in the *Potential value* field.



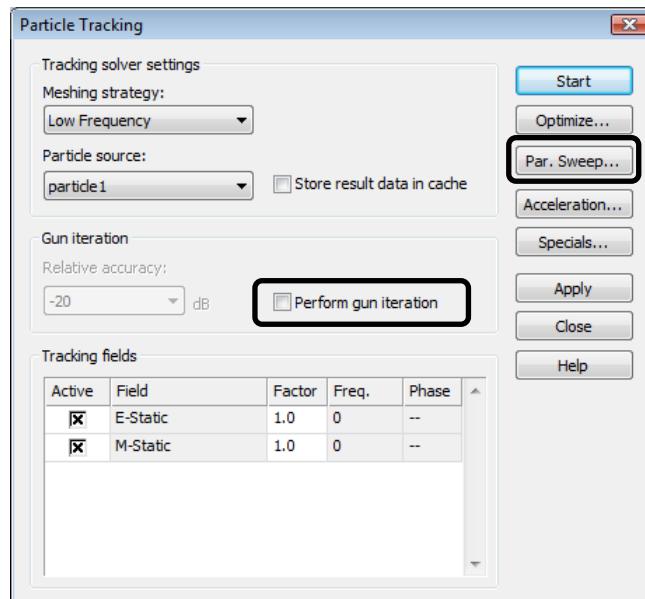
If you click the *OK* button, you will be asked to delete the current results. Just click the *OK* button and delete the results. Then the dialog box *New Parameter* opens to define the value of your parameter "phi".



Enter a value of $-3e4$ and click the **OK** button. You have successfully defined your first parameter. The values of your parameter can be edited and checked in the *Parameter List* window, that is usually located in the lower left part of the main window:

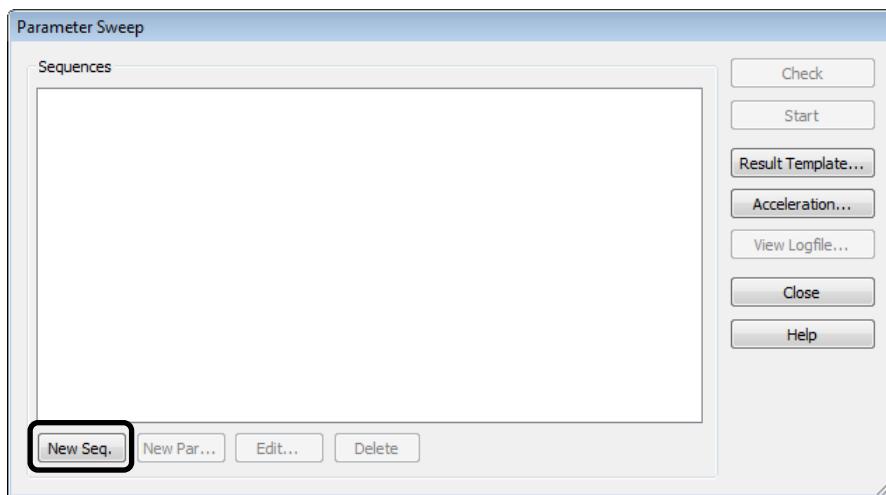
Name	Expression	Value	Description
phi	$= -3e4$	-30000	
<new variable>			

Since we did not change the value of the cathode's potential, the results of the simulation would be the same. We will now change the setup to run a so called *Parameter Sweep* to get the emitted current for potentials in the range from -32 to -28 kV. To do this open the *Particle Tracking* solver dialog box *Simulation: Solver \Rightarrow Setup Solver*.

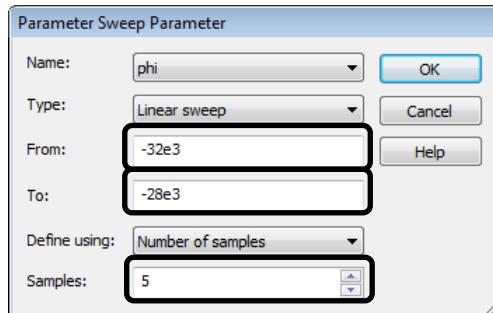


To save some time during the parameter sweep disable the checkbox *Perform gun iteration*. The tracking solver will now run only one calculation and will not operate in the iterative mode.

Click the button *Par. Sweep* to open the dialog box *Parameter Sweep* and to configure the parameter range and also the expected results of the parameter sweep.

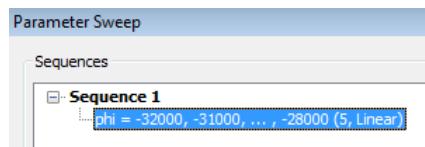


In this dialog box you can specify calculation sequences that consist of various parameter combinations. To add such a sequence, click the New Seq. button now. Then click the New par button to add a parameter variation to the sequence:

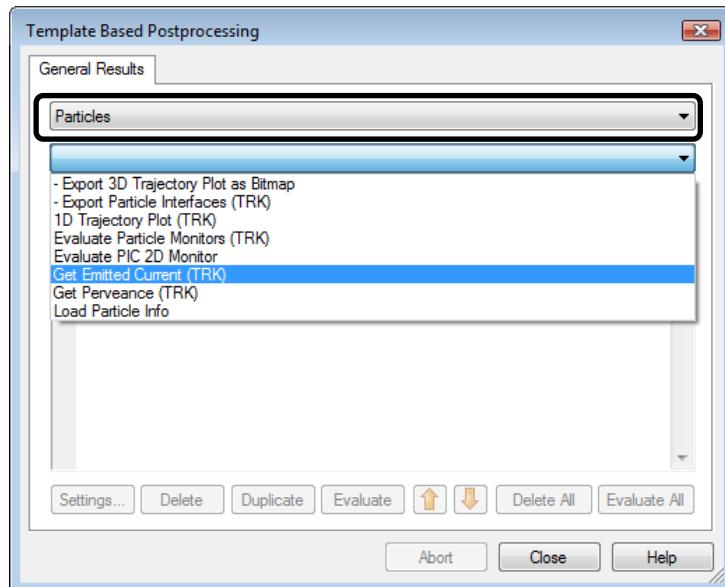


In the resulting dialog box you can select the name of the parameter to vary in the *Name* field. Then you can specify different sweep types to define the sampling of the parameter space (*Linear sweep*, *Logarithmic sweep*, *Arbitrary points*). Depending on this selection the sampling can be defined further, e.g. the linear sweep option allows us to define the lower (*From*) and upper (*To*) bounds for the parameter variation as well as the definition of either the number of samples or the step width.

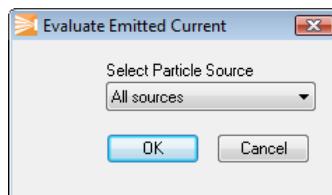
In this example you should perform a linear sweep from -32 kV to -28 kV in 5 steps. Click the OK button to confirm your changes. The definition of the sequence is finished but we still need to configure the expected result, the emitted current. The parameter sweep dialog box should look as follows:



Next, you have to specify the results that you are interested in. Therefore click on the button *Result Template*. The *Template Based Postprocessing* dialog box opens. Templates are separated into several *Template Groups*.



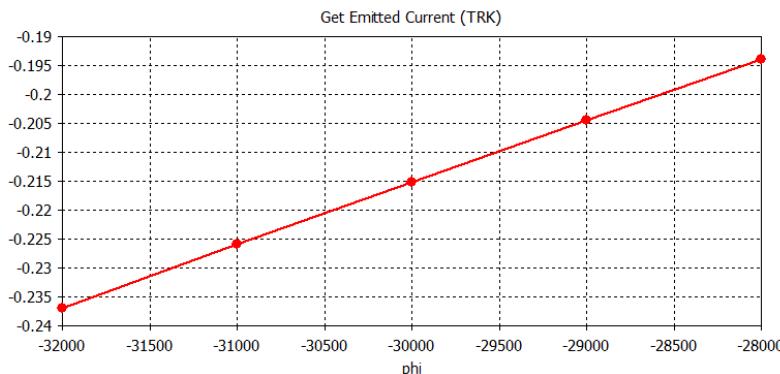
Choose the template *Get Emitted Current (TRK)* in the *Particles* group. This template evaluates and records the emitted current of the particle source. Once you choose this macro, a dialog box opens where the name of the particle source to be evaluated, is entered.



Since only one particle source is defined, the setting “All sources” evaluates only this defined source. Keep the default value and click *OK* to close this window. The *Template Based Postprocessing* dialog box should be still open and contain the following row:

	Result name	Type	Template name	Value
1	Get Emitted Current (TRK)	0D	Get Emitted Current (TRK)	

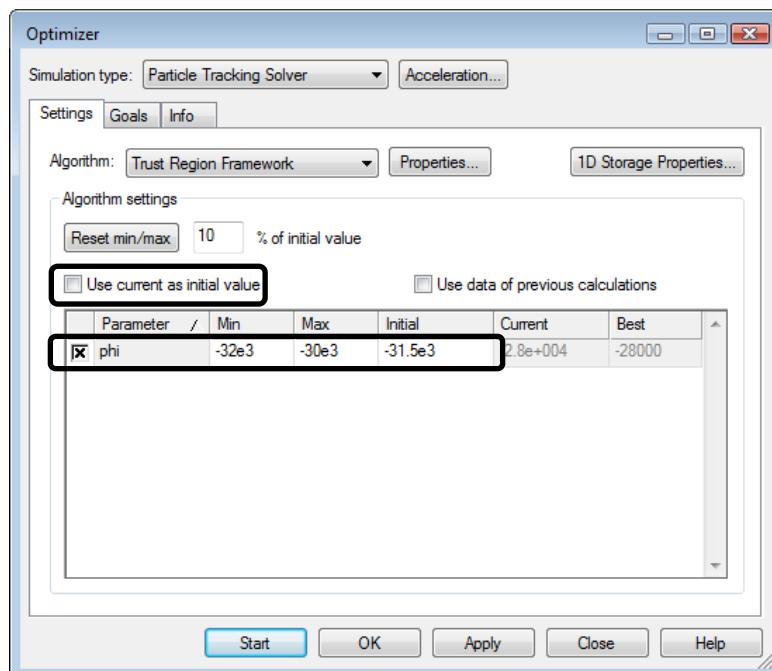
Click the *Close* button to return to the parameter sweep definition via *Simulation: Solver* \Rightarrow *Par. Sweep* . Now start the parameter sweep by clicking the *Start* button. The calculation may take a few minutes. After the solver has finished its work, leave the dialog box by clicking the *Close* button. The navigation tree contains a new item called *Tables* from which you can select the item *NT: Tables* \Rightarrow *0D Results* \Rightarrow *Get Emitted Current (TRK)*. The 1D result plot should look like in the picture below and gives you the relation between input voltage and emitted current of the electron gun:



Automatic Optimization of the Structure

Let us assume that you wish to adjust the emitted current to a value of -0.22 A (which can be achieved within the parameter range of -32 kV to -30 kV according to the parameter sweep). Figuring out the proper parameter may be a lengthy task that can also be performed automatically.

CST PARTICLE STUDIO offers a very powerful built-in optimizer feature for such parametric optimizations.

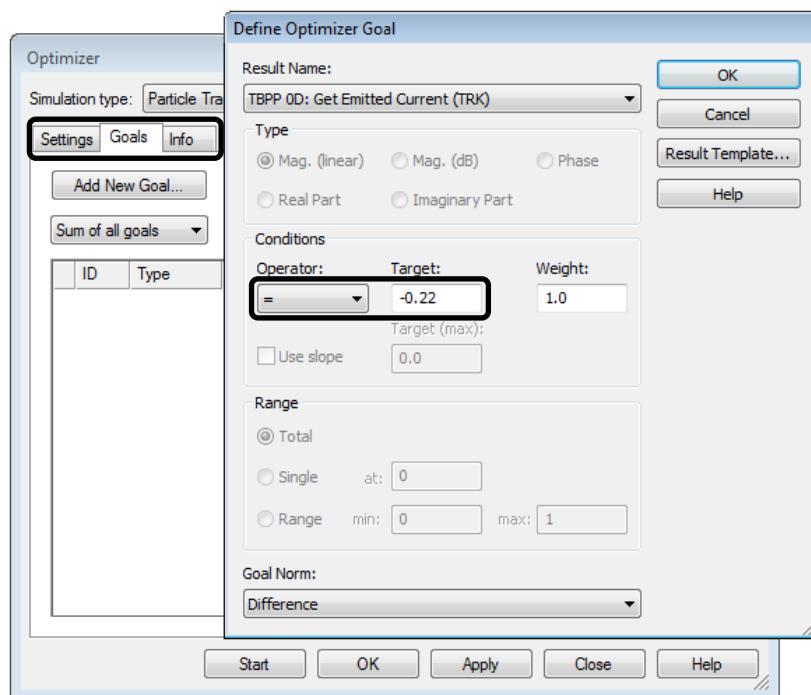


To use the optimizer, open the tracking solver control dialog box *Simulation: Solver* \Rightarrow *Setup Solver* in the same way as before, or directly via *Simulation: Solver* \Rightarrow *Optimizer* .

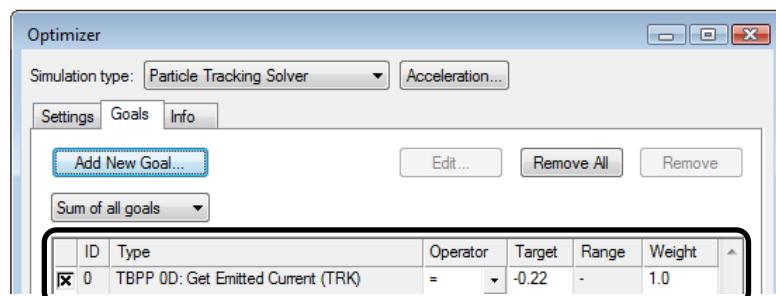
Click the *Optimize* button to open the optimizer control dialog box.

First activate the desired parameter(s) for the optimization in the *Parameters Tab* of the optimization dialog box, here the parameter "phi" should be checked. Next specify the minimum and maximum values for this parameter during the optimization. From the parameter sweep, we already know that the searched potential is greater than -32 kV and lower than -30 kV. Therefore you can enter a parameter range between -32 kV and -30 kV. Deactivate *Use current as initial value* and set the initial start value for the optimization, e.g. to -31.5 kV.

For this simple example, the other settings can be kept as default. Refer to the online documentation for more information on these settings. You can specify a list of goals you wish to achieve during the optimization. In this example the objective is to find a parameter value for which the emitted current becomes -0.22 A. The next step is to specify this optimization goal. Switch to the *Goals Tab* and click *Add New Goal*.

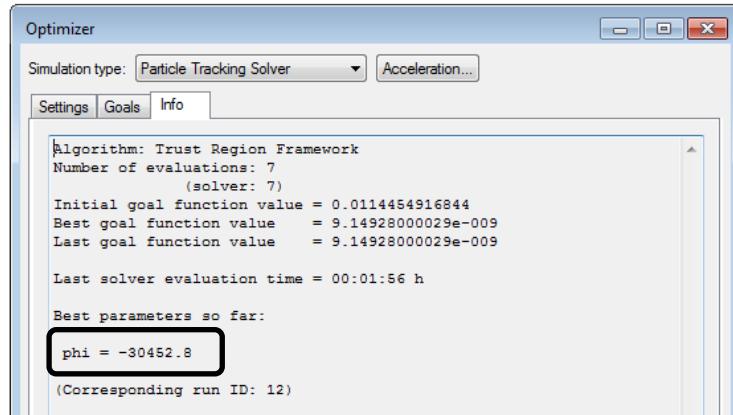


Now you can define the goal for the emitted current. Since you would like to find a value of -0.22 A, you should select the equal operator in the conditions frame. Then set the Target to -0.22. After you click OK, the optimizer dialog box should look as follows:



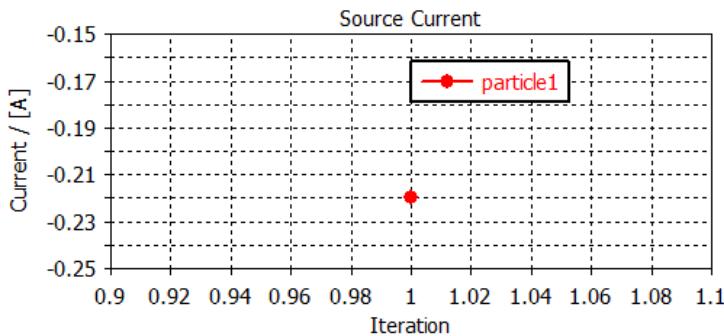
Note: The optimizer is capable of optimizing multiple parameters at once. Detailed information can be obtained from the online help.

Up to now, you have specified which parameters to optimize and set the goal that you want to achieve. The next step is to start the optimization procedure by clicking the *Start* button. As shown in the next picture, the optimizer will display its progress in an output window in the *Info* tab which is activated automatically. When the whole process is finished, confirm by saving the new parameter settings. The optimizer output window contains the best parameter values in order to achieve the desired goal.



Note that due to sophisticated optimization technology only seven solver runs are necessary to find the optimal solution with very high accuracy.

Click the *Close* button to leave the dialog box. Now look at the final result of the emitted current for the optimal parameter setting $\phi = -30348.6$ V by clicking *NT: 1D Results* \Rightarrow *Particle Source Current*. You should obtain the following curve:



As you can see, the final amount of emitted current for the optimized voltage parameter is -0.22 A as it was previously defined by the setting of the optimization goal.

Additional Information: The Particle Tracking Specials Dialog Box

The specials dialog box offers further options to change the solver properties. Open this dialog box by selecting *Simulation: Solver* \Rightarrow *Setup Solver*  and click the *Specials* button.



The *Tracking* frame offers the possibility to change specific settings of the particle tracking algorithm. The setting *Max. timesteps* defines the maximum number of simulated steps performed by the tracking algorithm. The *Min. pushes per cell* value determines the spatial sampling rate of the particle trajectories. The *Timestep dynamic* parameter specifies the variation of the time step between two pushes and introduces a dynamic adaptation of the time step to the highest particle velocity. Activating the checkbox *Monitor charge/current* results in monitoring the space charge and current density generated by the particles.

In the *Gun iteration* frame, the maximum number of iterations of consecutive electrostatic simulations and particle tracking computations is defined. The *Relaxation* parameter describes the influence of the last obtained space charge distribution to the overall charge distribution which is considered in the next electrostatic computation.

In the *Sampling* section, the sampling rate of calculated time steps and the sampling rate of the particles for the monitored particle tracking result are defined.

The *Periodic boundaries* and *Open boundaries* frames allow you to define alternative boundaries for field solvers which cannot handle these types of boundaries.

Summary

This example gave you a basic overview of the key concepts of the tracking solver of CST PARTICLE STUDIO. You should now have a basic idea of how to do the following:

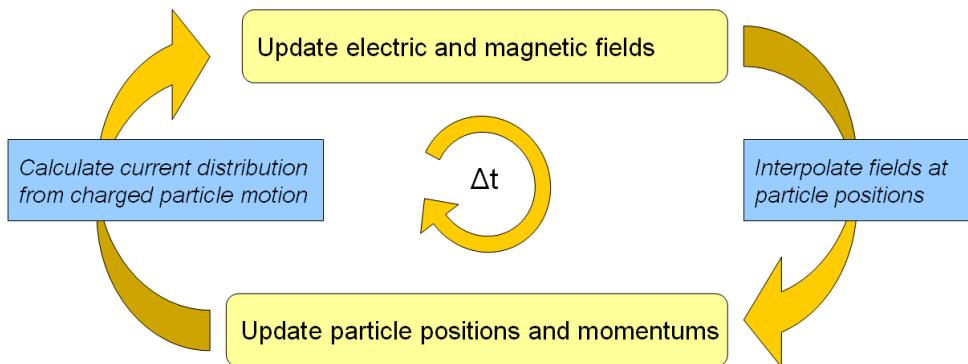
1. Create a structure using the solid modeler
2. Specify the solver parameters, check and modify the mesh and start the tracking simulation
3. Visualize the electromagnetic field distributions and the particles' trajectory
4. Define a structure using parameters
5. Use the parameter sweep tool for parameter studies
6. Perform automatic optimizations

If you are familiar with all these topics, you have a very good starting point for improving your usage of CST PARTICLE STUDIO.

For more information on a particular topic, we recommend that you look at the contents page of the online help manual, which can be opened via *Help ⇔ CST STUDIO SUITE – Help*. If you have any further questions or remarks, do not hesitate to contact our technical support team. We also strongly recommend that you participate in one of our special training classes held regularly at a location near you. Please ask us for details.

Simulation Workflow: Particle-in-Cell

The basic procedure of running the particle-in-cell (PIC) solver is very similar to the one demonstrated in the tracking simulation workflow. In contrast to the tracking solver the PIC solver simulates particles in self-consistent fields by using a time integration scheme for particles and electromagnetic fields:



The algorithm shown above reflects the fundamentals for the PIC method. In contrast to the CST tracking and wakefield solvers, the interdependency of fields and charges is taken into account.

The following example demonstrates how to perform a PIC calculation for a simple output cavity of a klystron. Studying this example carefully will allow you to become familiar with many standard operations that are necessary to perform a PIC simulation within CST PARTICLE STUDIO.

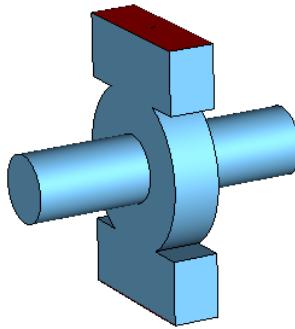
Go through the following explanations carefully even if you are not planning to use the software for PIC simulations. Only a small portion of the example is specific to this particular application type since most of the considerations are general to all solvers and application domains.

The following explanations always describe the menu-based way to open a particular dialog box or to launch a command. Whenever available, the corresponding toolbar item is displayed next to the command description. Due to the limited space in this manual, the shortest way to activate a particular command (i.e. by pressing a shortcut key or activating the command from the context menu) is omitted. You should regularly open the context menu to check available commands for the currently active mode.

The Structure

This workflow example demonstrates how to build up the output cavity of a klystron for a PIC simulation. A klystron is a device to amplify microwave and/or radio frequency signals. The output resonator is the last stage (cavity) of a klystron. The amplified signal can be extracted by waveguide ports.

Since only the output resonator as a part of the klystron is simulated, a Gaussian emission model is used to define an already bunched particle beam.

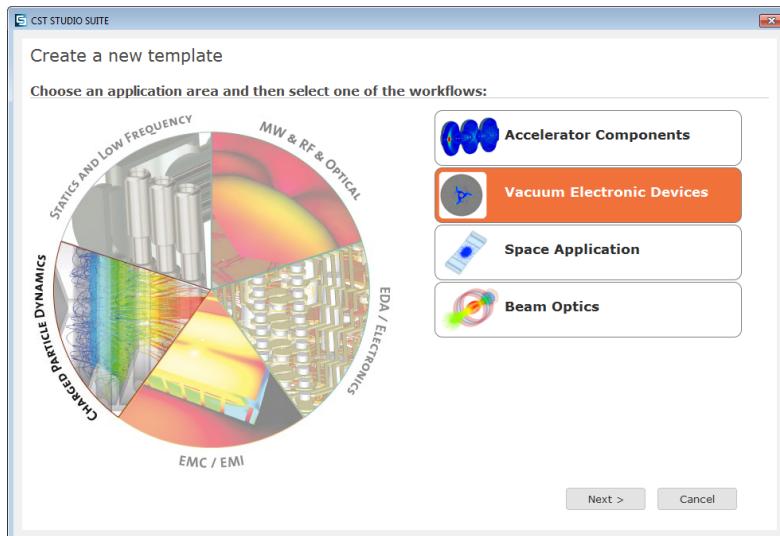


CST PARTICLE STUDIO allows you to define the properties of the background material. Anything you do not fill with a particular material will automatically be considered as the background material. For this structure, it is sufficient to model only the vacuum space. The background material will be set to perfect electrical conductor (PEC).

Create a New Project

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

Next you should choose the application area, which is *Charged Particle Dynamics* for the example in this tutorial and then select the workflow by double-clicking on the corresponding entry.

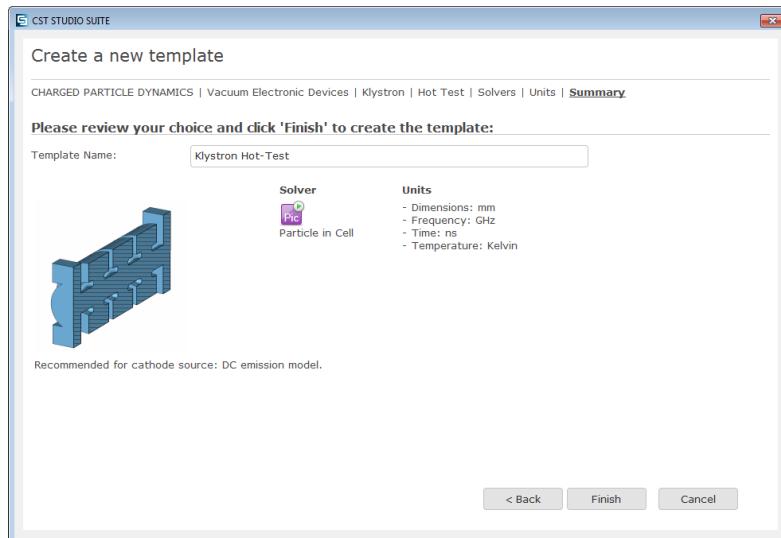


Please select then the following workflow: *Vacuum Electronic Devices* \Leftrightarrow *Klystron* \Leftrightarrow *Hot Test* \Leftrightarrow *Particle in Cell* .

At last you are requested to select the units which fit your application best. For this example, please select the dimensions as follows:

Dimensions:	mm
Frequency:	GHz
Time:	ns
Temperature:	Kelvin

For the specific application in this tutorial the other settings can be left unchanged. After clicking the *Next* button, you can give the project template a name and review a summary of your initial settings:



Finally click the *Finish* button to save the project template and to create a new project with appropriate settings. CST PARTICLE STUDIO will be launched automatically due to the choice of this specific project template within the application area *Charged Particle Dynamics*. Save the newly created “Untitled” project on your hard disk using a name of your choice.

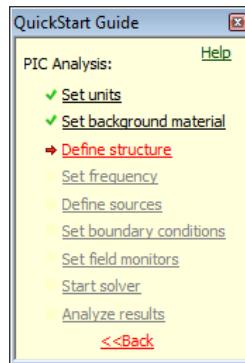
Please note: When you click again on the *File: New and Recent* you will see that the recently defined template appears below the *Project Templates* section. For further projects in the same application area you can simply click on this template entry to launch CST PARTICLE STUDIO with useful basic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

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

Open the PIC QuickStart Guide

An interesting feature of the online help system is the QuickStart Guide, an electronic assistant that will guide you through your simulation. If it does not show up automatically, you can open this assistant by selecting *QuickStart Guide* from the Help button  in the upper right corner.

The following dialog box should then be visible at the upper right corner of the main view:



As the project template has already set the solver type, units and background material, the *PIC Analysis* is preselected and some entries are marked as done. The red arrow always indicates the next step necessary for your problem definition. You do not have to follow the steps in this order, but we recommend you follow this guide at the beginning to ensure that all necessary steps have been completed.

Look at the dialog box as you follow the various steps in this example. You may close the assistant at any time. Even if you re-open the window later, it will always indicate the next required step.

If you are unsure of how to access a certain operation, click on the corresponding line. The *QuickStart Guide* will then either run an animation showing the location of the related menu entry or open the corresponding help page.

Define the Units

The *Klystron Hot-Test* template has already made some settings for you. The defaults for this structure type are geometrical units in mm and times in ns. You can change these settings by entering the desired settings in the units dialog box (*Home: Settings \Rightarrow Units*), but for this example you should just leave the settings as specified by the template. Additionally, the used units are also displayed in the status bar:



Define the Background Material

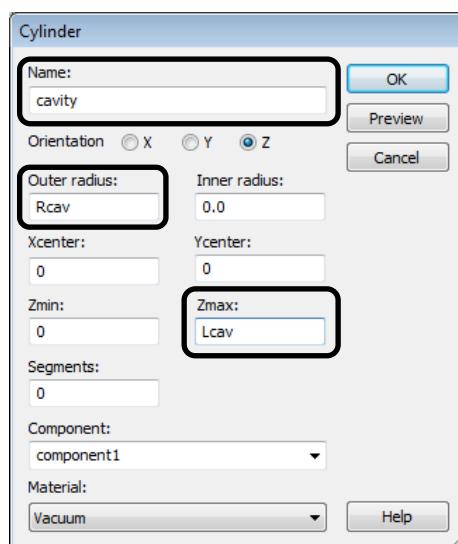
As discussed above in the Structure section, the klystron cavity is surrounded by perfect electrical conductor (PEC). The material type *PEC* is already set as default background material in the *Klystron Hot-Test* template. You may change the background material in the corresponding dialog box (*Simulation: Settings \Rightarrow Background*). For this example no change of the background material is needed.

Model the Structure

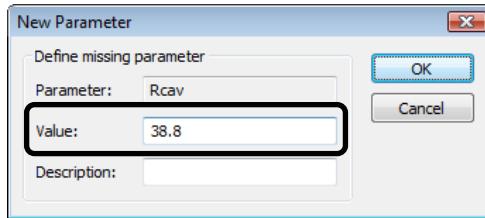
Having defined the initial general settings, the 3D structure view is now visible and the working plane is shown therein. The working plane can be turned off (and on) by clicking on *View: Visibility \Rightarrow Working Plane* .

Then, you can start building the 3D structure. First, create a vacuum cylinder along the z-axis of the coordinate system using the following steps:

1. Select the cylinder creation tool *Modeling: Shapes \Rightarrow Cylinder* .
2. Press the *ESC* key to open the dialog box. Do not click a point in the working plane.
3. Enter "cavity" as name.



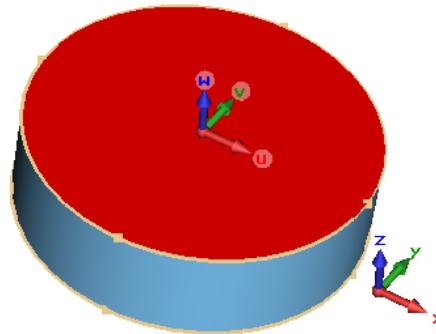
4. Enter the parameters "Rcav" as outer radius and "Lcav" as Zmax. Click the OK button to confirm the changes.
5. The "New Parameter" dialog box appears. Enter 38.8 as value for Rcav. Press the *Return* key to confirm. It is also possible to add a description of the parameter.



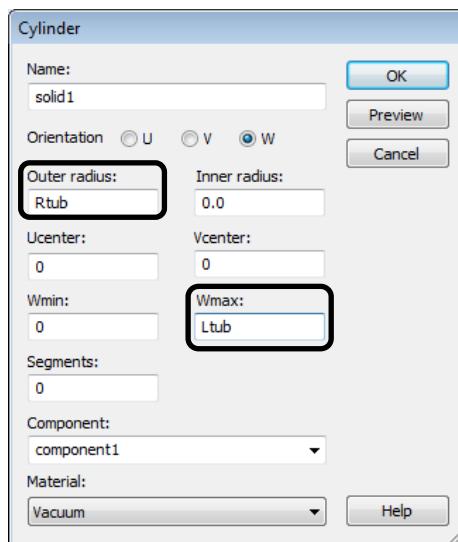
6. Another "New Parameter" dialog box appears. Enter 22 as value for Lcav. Press the *Return* key to confirm. The defined parameters are shown in the Parameter List window of the CST STUDIO SUITE.

	Name	Expression	Value	Description	Type
	Lcav	= 22	22		None ▾
	Rcav	= 38.8	38.8		None ▾
<i>(new variable)</i>					

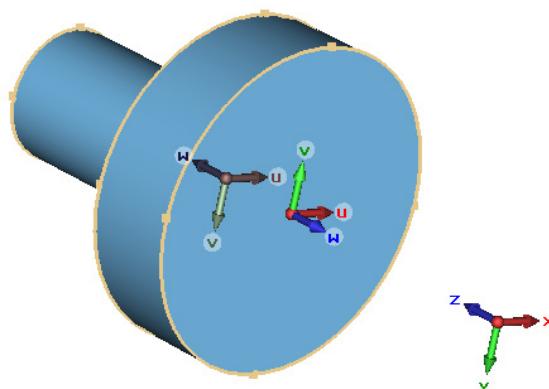
7. Activate and move the working coordinate system to the center of the upper cylinder face: Select *Modeling: WCS* \Rightarrow *Align WCS* and pick the face in the maximum z-direction. This setting is used in the following step (step 8) to define a vacuum cylinder based on the axes of the working coordinate system.



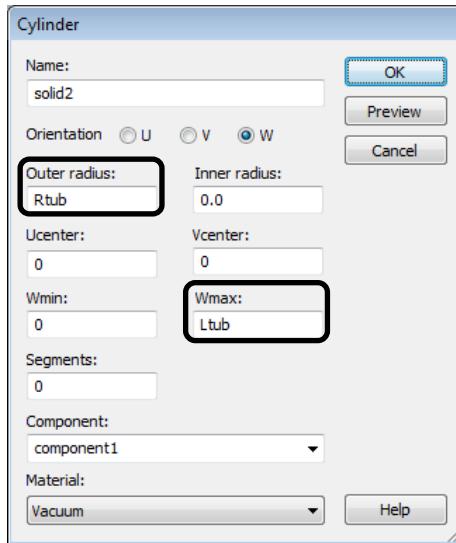
8. Define a second vacuum cylinder: select the cylinder creation tool *Modeling: Shapes* \Rightarrow *Cylinder* . Press the *ESC* key to open the dialog box.



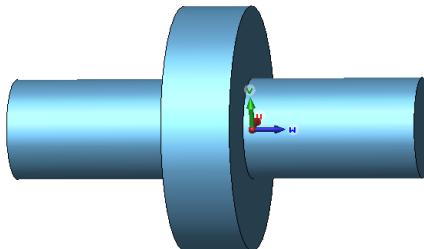
9. Enter the parameters "Rtub" as outer radius and "Ltub" as Wmax. Press the *Return* button to confirm the changes.
10. The "New parameter" dialog boxes appear again. Choose 15.9 for Rtub and 55 for Ltub. Press the *Return* button to confirm.
11. In the same way as before move the origin of the working coordinate system to the center of the lower face of the cavity cylinder. Select *Modeling: WCS* \Rightarrow *Align WCS* and pick the face in the minimum z-direction.



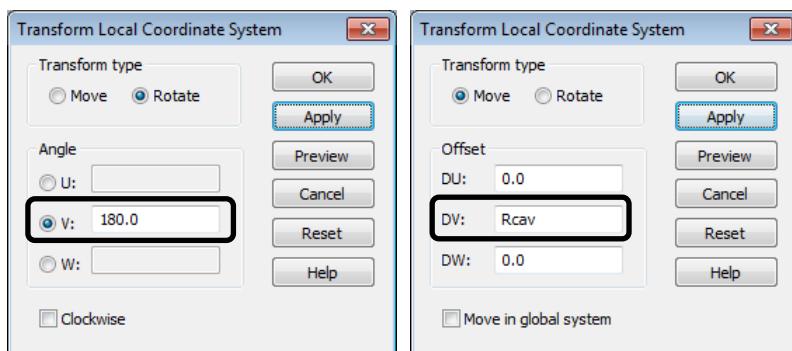
12. Define a third vacuum cylinder. Select the cylinder creation tool *Modeling: Shapes* \Rightarrow *Cylinder* . Press the *ESC* key to open the dialog box.
13. Enter the parameters "Rtub" as outer radius and "Ltub" as Wmax. Press the *Return* button to confirm the changes.



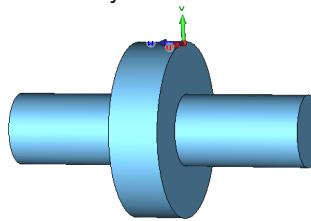
In the 3D structure view, the structure below should be visible now:



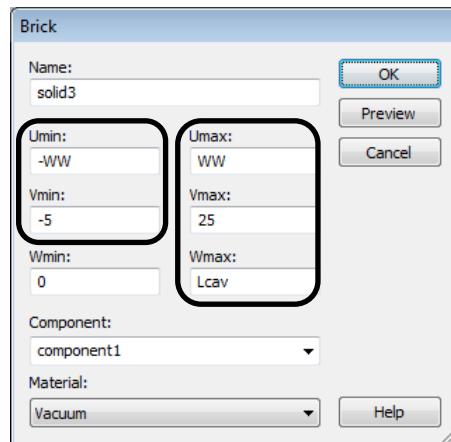
14. Rotate the local coordinate system 180° around the v-axis: open the Transform dialog box from *Modeling: WCS \Rightarrow Transform WCS* and select the *Rotate* button. Enter 180° for the V-direction. Then click the *Apply* button.
15. Move the local coordinate system about Rcav in v-direction: Select the *Move* button and enter Rcav for the DV shift. Click *OK* to confirm.



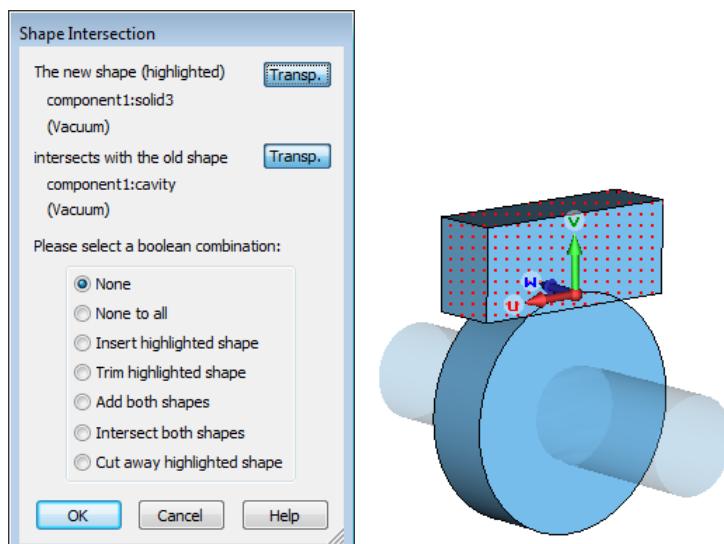
The origin of the local coordinate system should now be shifted to this position:



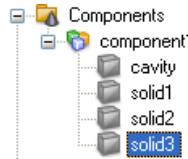
16. Define a vacuum brick. Select the brick creation tool *Modeling: Shapes* \Rightarrow *Brick* . Press the *ESC* key to open the dialog box.



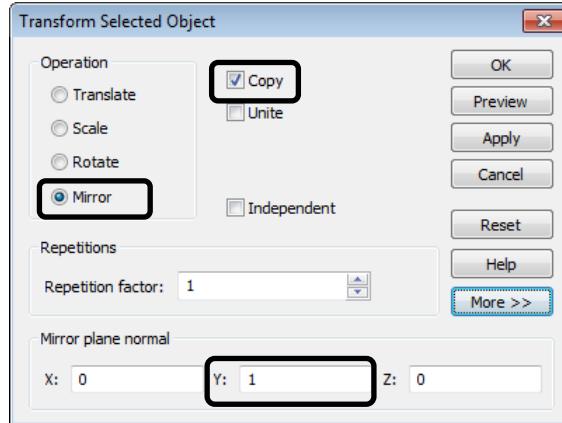
17. Enter the values as shown in the picture above. For Umax enter the length "WW" and for Umin the length "-WW". Click the *OK* button. The "New Parameter" dialog box will appear again. Enter 36.1 as length for WW and click *OK*.
18. Since the structures intersect, the "Shape Intersection" dialog box shown below appears. Select "None" and click *OK*.



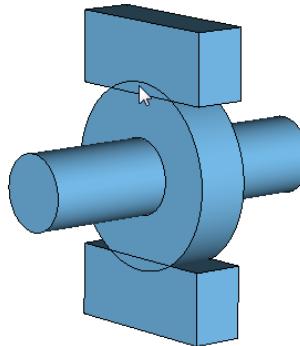
19. Switch to the global coordinate system by disabling the WCS: *Modeling: WCS* \Leftrightarrow *Local Coordinate System* .
20. Select "solid3" in the navigation tree.



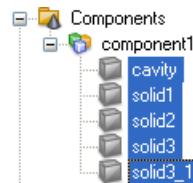
21. Open the "Transform Selected Object" dialog box: *Modeling: Tools* \Leftrightarrow *Transform* .



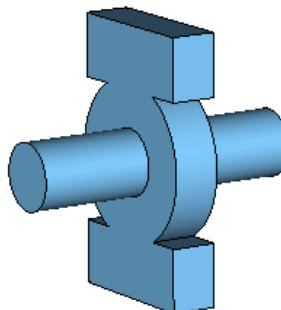
22. Enable "Mirror" and "Copy". Choose "Y" as mirror plane normal. Click the OK button.
23. Since the structures intersect, the "Shape Intersection" dialog box appears again. Select "None" and click OK. Your structure should now look like this:



24. Select all existing solids in the navigation tree. Transform all selected solids into one vacuum solid: *Modeling: Tools* \Leftrightarrow *Boolean* \Leftrightarrow *Add* .



25. Finally the structure should look like this:

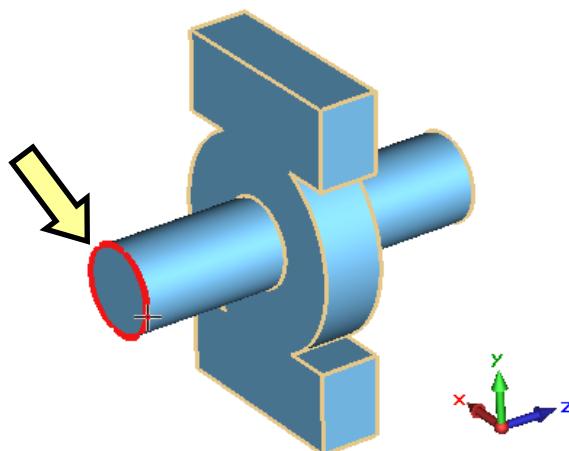


Congratulations! You have just created your first PIC structure within CST PARTICLE STUDIO.

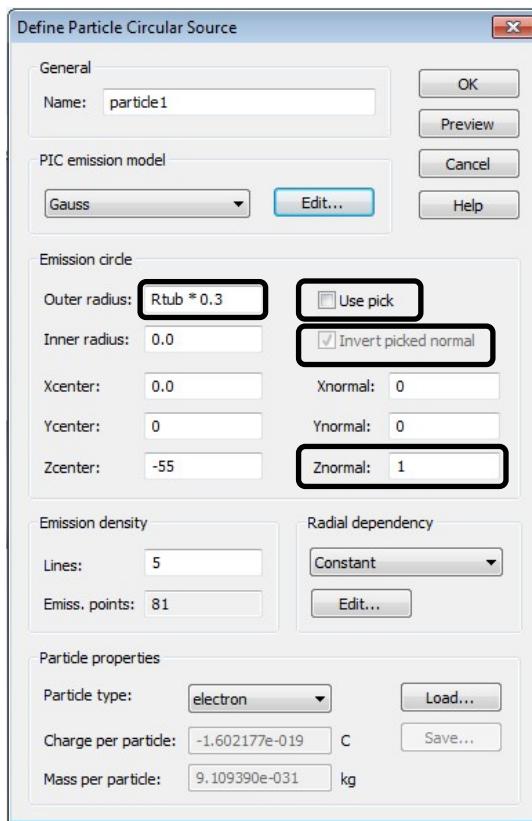
Define the Particle Source

We use an electron source as particle source. The emission is based on a Gaussian emission model. Since the beam has a circular cross section, the circular particle source  can be used.

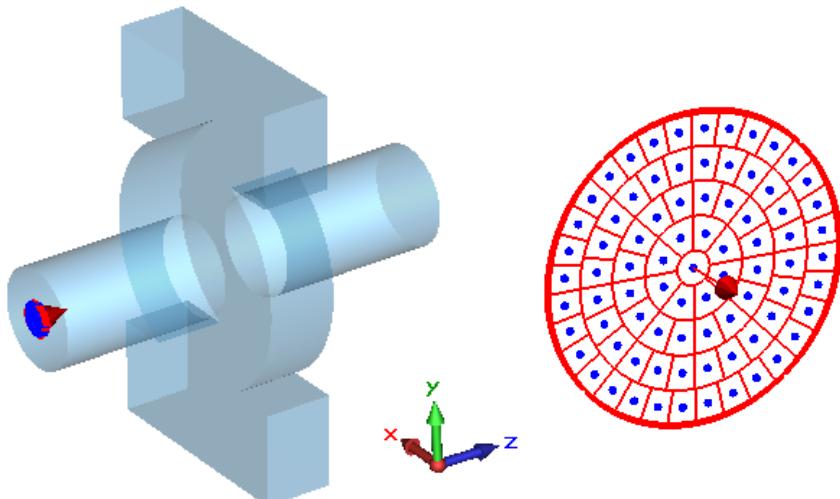
1. Define a circular particle source on the beam tube at the lower z-coordinate:
Simulation: Sources and Loads \Rightarrow *Particle Sources* \Rightarrow *Particle Circular Source* 
Select the following edge (lower z-direction) of the beam tube with a double-click:



2. The dialog box "Define Particle Source on Circle" opens where you can modify the settings of the particle source:

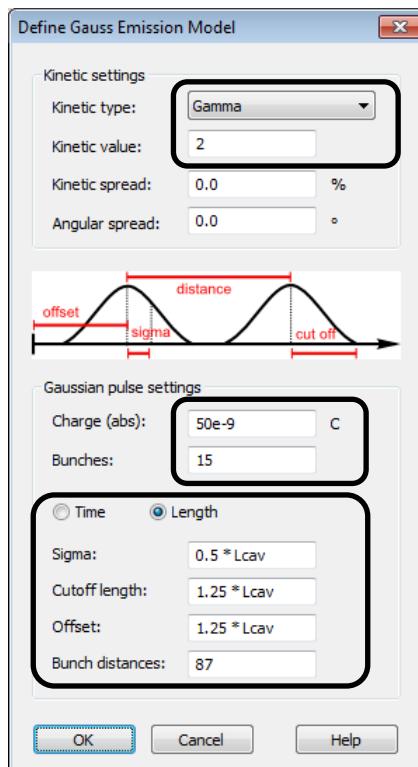


3. Select the checkbox "Invert picked normal", deselect the checkbox "Use pick", enter a *Outer Radius* value of $R_{tub} \times 0.3$ and a *Znormal* value of 1. Click the Preview button to check these settings.



4. In the PIC emission model section, the *Gauss* emission model is already selected from the drop-down list. Click the *Edit* button to define the parameters of the Gaussian emission model. The *Define Gauss Emission Model* dialog box should open:

Note: For more information about emission models and appropriate settings please refer to the online manual.



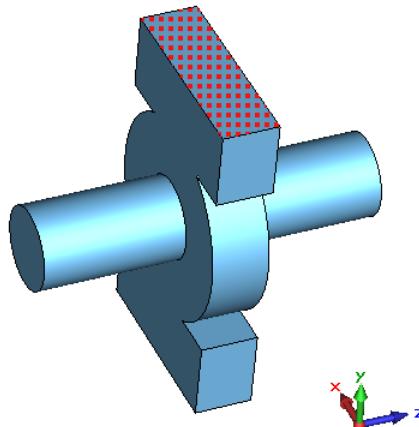
5. Enter the values shown in the following table and click *OK* to confirm the changes. Click *OK* again to close the "Define Particle Circular Source" dialog box.

Setting	Value
Kinetic type	Gamma
Kinetic value	2
Charge (abs)	50e-9
Bunches	15
Time / Length	Length
Sigma	0.5*Lcav
Cutoff Length	1.25*Lcav
Offset	1.25*Lcav
Bunch distance	87

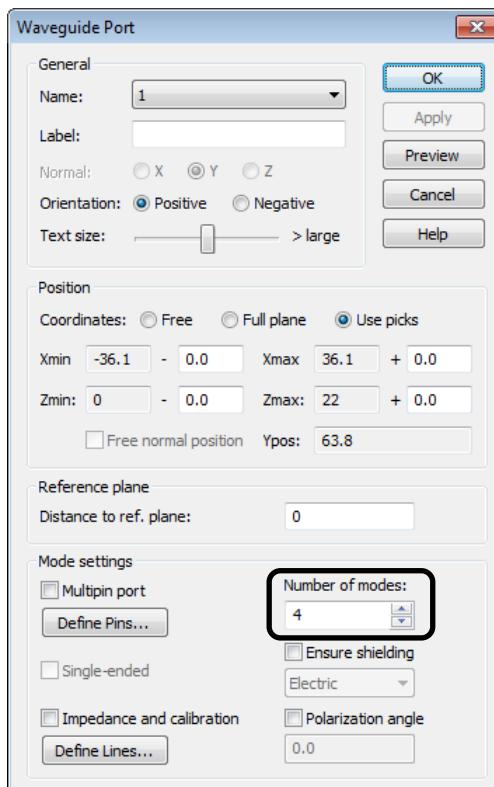
Define the Ports

In our example waveguide ports are used to extract the energy the cavity. The ports are not used for excitation.

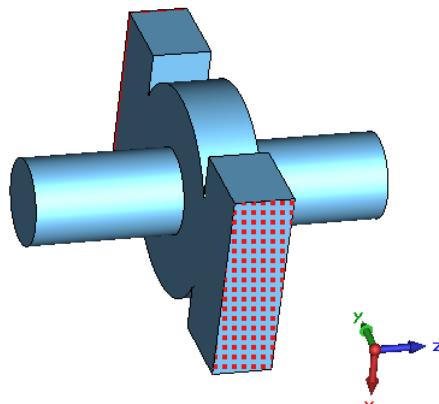
1. Pick the following face of one brick and double-click to define a port on it: *Modeling: Picks \Rightarrow Pick Points, Edges or Faces*.



2. Open the "Waveguide Port" dialog box to define a waveguide port on the picked face (upper y-direction): *Simulation: Sources and Loads \Rightarrow Waveguide Port*.



3. Change the number of port modes to 4 and click *OK* to confirm.
4. For the other brick define a port on the opposite face (lower y-direction) in the same way: Pick the face, enter the Waveguide Port dialog box and change the *Number of modes* to 4.

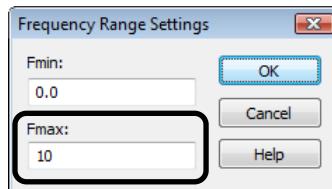


5. Confirm the settings with the *OK* button. The port definition is finished now.

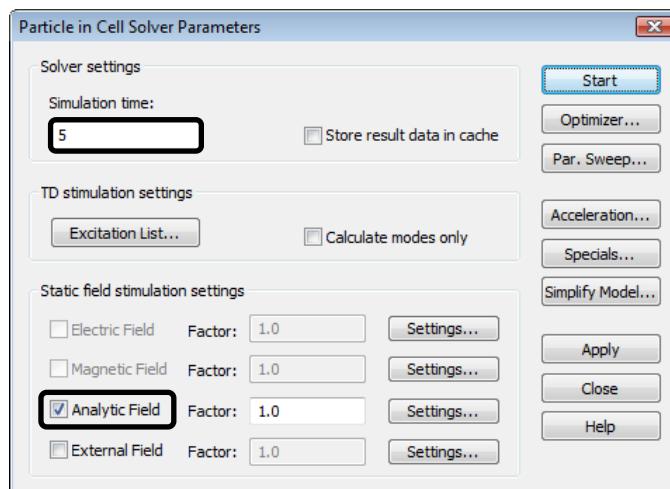
Simulation Setup

The solver parameters can be set up within the PIC solver dialog box. A frequency range must be defined since the mesh is created with respect to the maximum frequency.

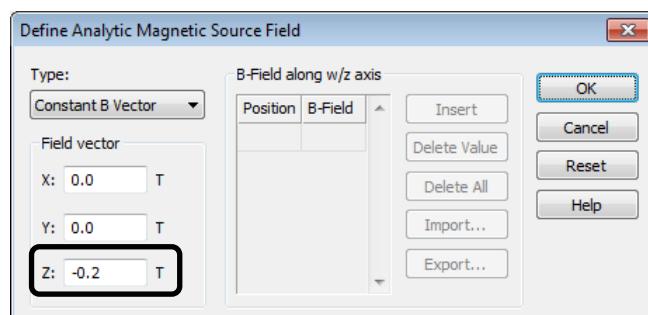
1. Define the maximum frequency within the *Frequency Range Settings* dialog box *Simulation: Settings \Rightarrow Frequency*.



2. Enter a frequency of 10 for Fmax and click the OK button.
3. Open the PIC solver dialog box: *Simulation: Solver \Rightarrow Setup Solver*.

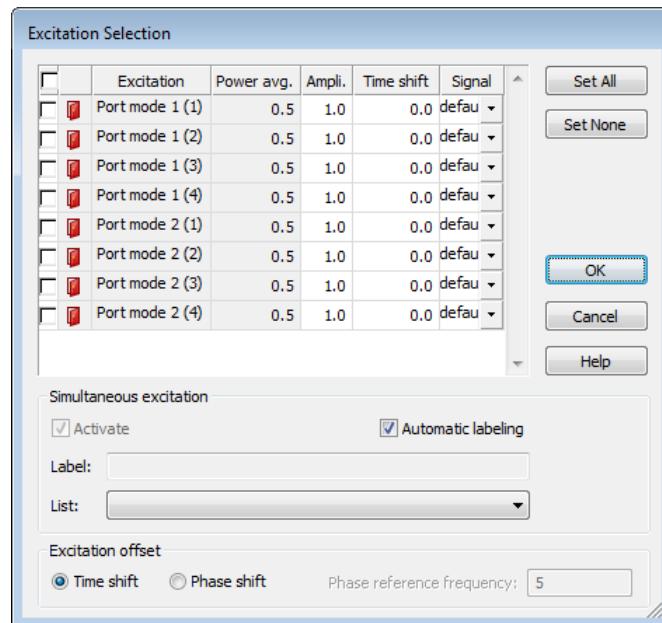


4. Change the simulation time to 5 and enable the checkbox *Analytic Field*.
5. To define the analytic field, click the *Settings* button of the analytic fields. The *Define Analytic Magnetic Source Field* dialog box appears.



6. Change the z-component of the "Constant B Vector" to -0.2 and click the *OK* button to confirm. This will apply a homogeneous magnetic flux density of 0.2T along the -z direction to focus the particle beam.

Before leaving the *Particle in Cell Solver Parameters* dialog box we want to draw your attention to the *Excitation List* button which might be important if ports or other HF-sources are defined:

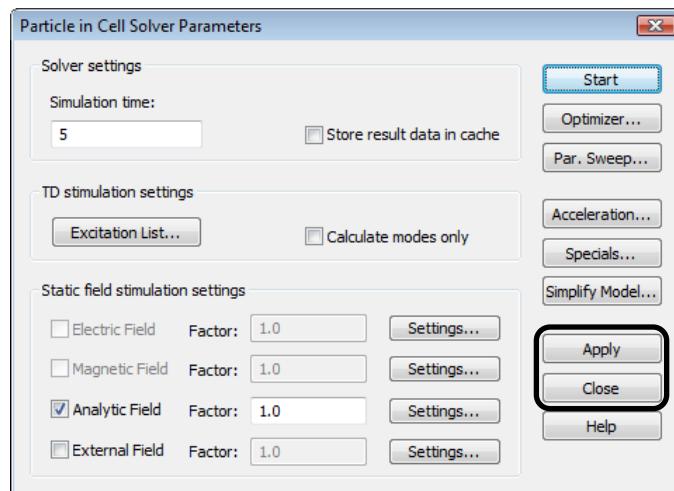


If the ports are excited, one can define the amplitude and the time shift for a previously defined excitation signal. For example, applications like traveling wave tubes feature driven ports.

Note: The amplitude value is the amplitude of the port signal (units sqrt(W)), which represents the square root of the peak power applied to the port. For simplicity, the corresponding average power of the exited port is shown in the column *Power avg.*

No changes need to be made for this example, so you can leave the dialog box by clicking *Cancel*.

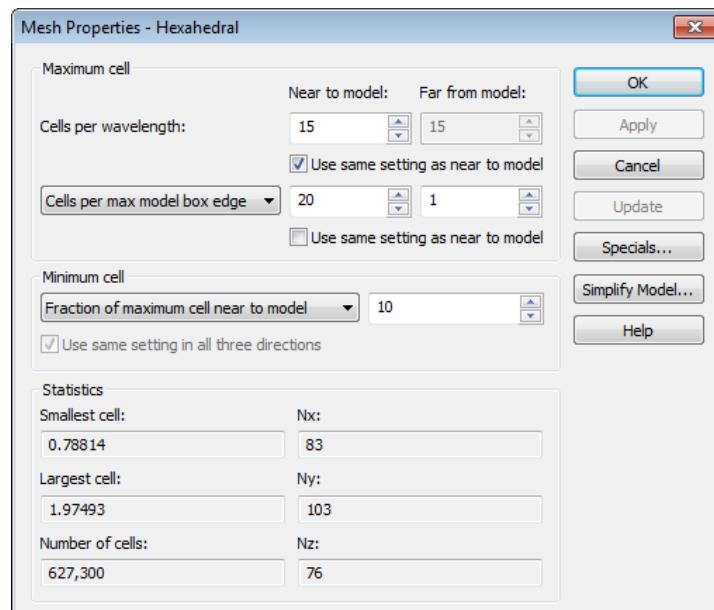
Now the solver could be started, but before the mesh will be modified and some particle monitors will be defined. First click the *Apply* and then the *Close* button in the main solver dialog box.



Refine the mesh

The mesh settings for this example are already specified in a good way by the *Klystron Hot-Test* template. However, in some cases the mesh has to be adjusted manually, as the mesh does not know anything about the particle movement. To change the mesh settings, proceed as follows:

1. Click on *Simulation: Mesh* \Rightarrow *Global Properties* to open the dialog box of the mesh properties.



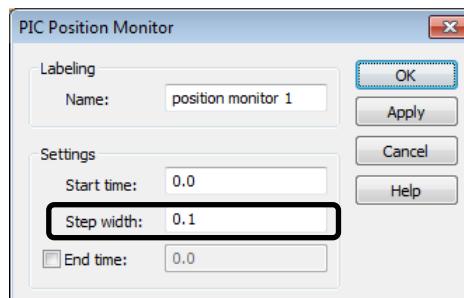
2. Play a little bit with the settings. E.g. set *Cells per wavelength – Near to model* to 20, click *Apply* to observe the change in the number of mesh cells.
3. Undo your changes and click *OK* to leave the dialog box.

Define Particle Monitors

To understand the interaction of particles with electromagnetic fields, it is often useful to gain an insight into the particle distribution. In this example, it may be interesting to see how particle bunches are deformed when moving through the beam tube.

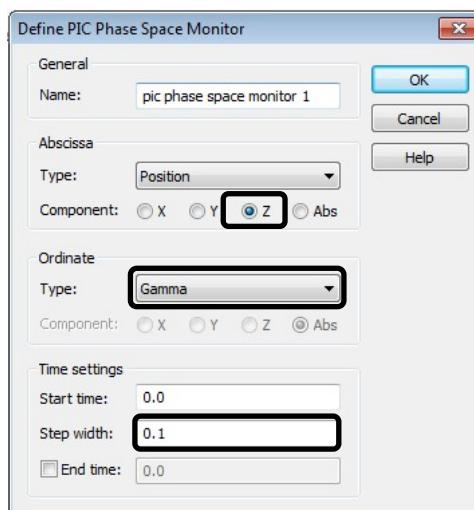
The particle distribution can be recorded with an equidistant sampling in time. You may need to switch back to the modeler mode by selecting the *Components* folder in the navigation tree before the monitor definition can be activated.

For this example a 3D PIC Position Monitor will be defined. Select and open the *PIC Position Monitor* dialog box: *Simulation: Monitors* \Rightarrow *PIC Position Monitor* .



Enter a *Step width* of 0.1 and create the monitor by pressing the *OK* button.

In addition to the 3D particle position monitor a phase space monitor will be set up. Select *Simulation: Monitors* \Rightarrow *PIC Position Monitor* \Rightarrow *PIC Phase Space Monitor*  to open the *PIC Phase Space Monitor* dialog box:



For the abscissa select the *z-position* and for the ordinate select *Gamma*. Enter a *Step width* of 0.1 for the time sampling. As the beam moves parallel to the z-axis, we are interested in monitoring the particle y as a function of the *z-position*.

Apart from the 3D position monitor and the phase space monitor, a 2D position monitor is available as well. Please refer to the online help for further details.

Start the Simulation

All necessary parameters have been now defined and you are ready to perform your first PIC simulation. You can start the solver directly by clicking *Home: Simulation*  *Start Simulation*. Alternatively, you can reopen the PIC solver dialog box, *Simulation: Solver*  *Setup Solver*, and start the solver by clicking the *Start* button.

In the progress window, a progress bar will be shown which informs you on the solver's status. Information regarding the operation will be displayed next to the progress bar. The most important stages are listed below:

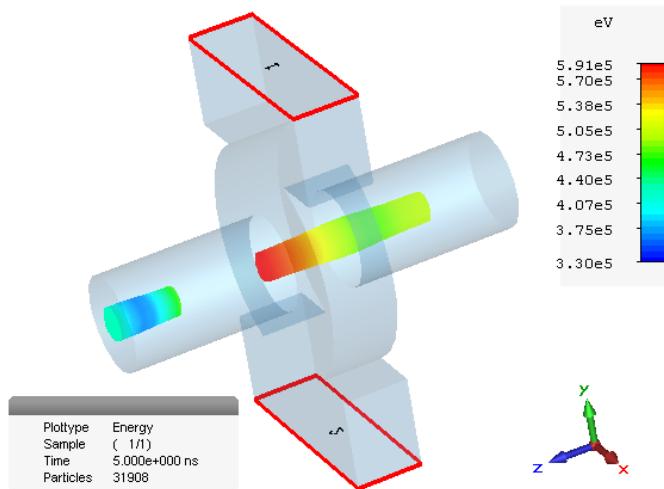
1. **Calculating matrices, preparing and checking model:** During this step, your input model is checked for errors such as invalid or overlapping materials.
2. **Calculating matrices, normal matrix and dual matrix:** During these steps, the system of equations, which will subsequently be solved, is set up.
3. **Transient analysis, calculating the port modes:** In this step, the solver calculates the port mode field distributions if any ports were defined. This information will be used later in the time domain analysis of the structure.
4. **Transient analysis, processing excitation / transient field analysis:** During this stage, the particles are emitted into the calculation domain and tracked through the electromagnetic fields. The solver stops after the previously defined *Simulation time* has been reached.

For this simple structure, the entire analysis takes a few minutes to complete.

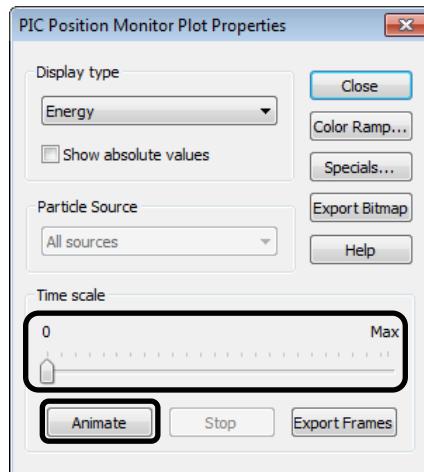
Note: During the simulation, the position of the particles can be watched by selecting *NT: 2D/3D Results*  *Particles*  *Particle preview* in the Navigation Tree. The view of particles can be then updated by pressing the F5 key or by clicking on *2D/3D Plot: Plot Properties*  *Update Results*.

Analyze the Simulation Results

The results of the PIC simulation can be analyzed in several ways. First, the charged particle motion can be visualized by selecting the result entry for the previously defined 3D particle monitor . Select *NT: 2D/3D Results* \Rightarrow *Particles* \Rightarrow *position monitor 1*.

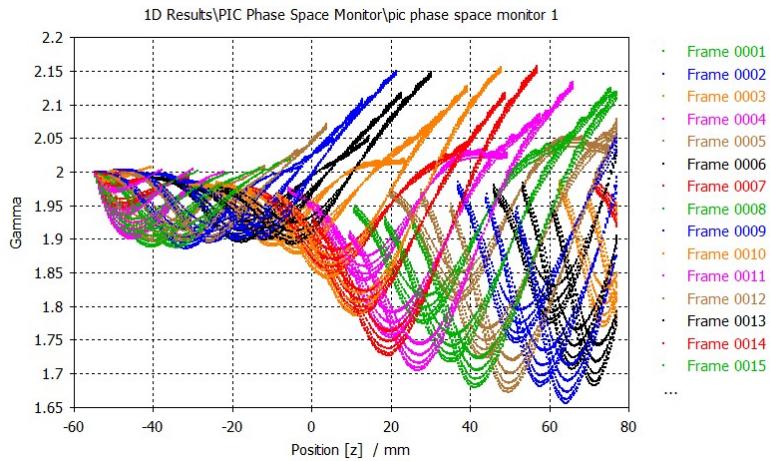


Probably you will see a transparent structure without any particles. The reason for this is that at $t = 0.0$ no particles have been emitted yet. Open the *PIC Position Monitor Plot Properties* dialog box by double clicking inside the 3D view window or by selecting *2D/3D Plot: Plot Properties* \Rightarrow *Properties*.

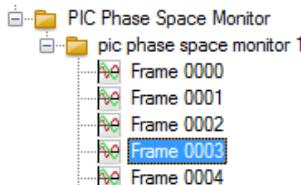


Use the slider to plot another time sample. Another way to move back and forth in the time sample sequence is to use the left and right arrow keys, after having clicked somewhere in the 3D view window. To start an automatic animation, click the *Animate* button in the *PIC Position Monitor Plot Properties* dialog box. This dialog box allows several other plot modifications, described in more detail in the online help. Close the dialog box by clicking the *Close* button.

The phase space plot monitor result can be accessed by selecting *NT: 1D Results* \Leftrightarrow *PIC Phase Space Monitor* \Leftrightarrow *pic phase space monitor 1*:

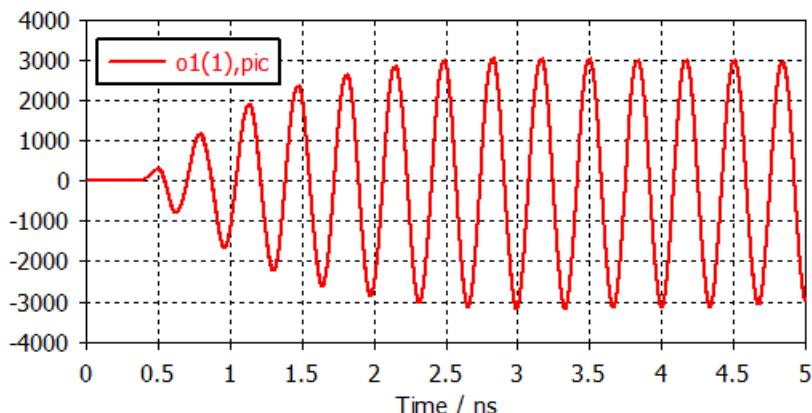


This result illustrates the gamma (proportional to energy) variation in time versus the longitudinal position. If you want to obtain the phase space at a specific instance in time you can select a single frame in the Navigation Tree:



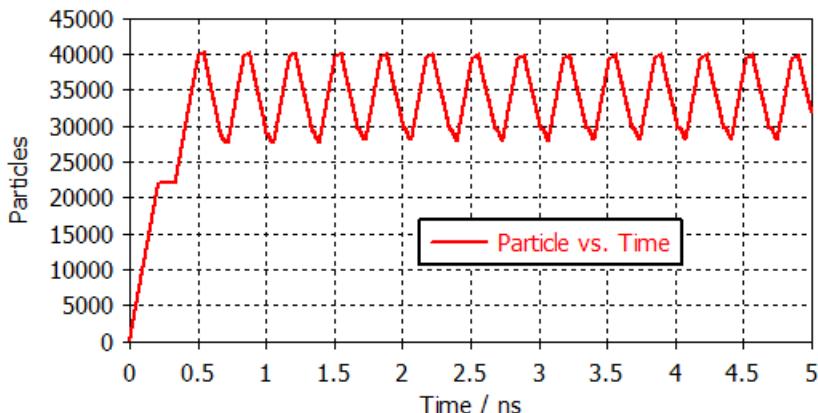
In addition to the results of the previously defined monitors, the PIC solver creates several other entries in the result tree. Below you can find a selection of interesting results:

Port Signals (*NT: 1D Results* \Leftrightarrow *Port signals*)



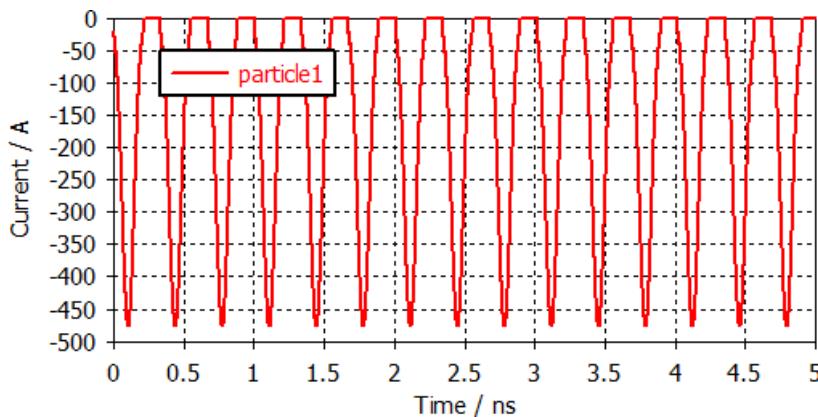
If ports are defined, the output signals at these ports are added automatically to the results. In this example the signal at port 1 shows that the bunched particle beam creates high power radio waves. As mentioned earlier, the output signals correspond to the square root of the peak power, which means that the average output power extracted from the beam amounts to $0.5 \times 3000 \times 3000$ W.

Particle Number (*NT: 1D Results* \Rightarrow *Solver Statistics [PIC]*)



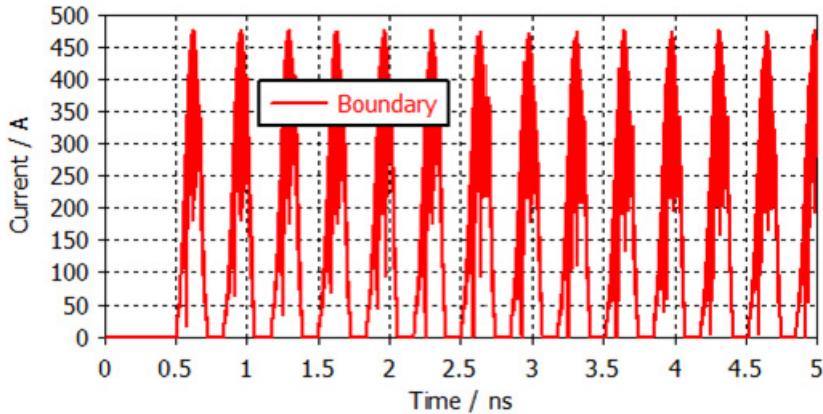
This 1D result shows the total number of macro particles inside the calculation domain vs. time. The curve increases when new particles are emitted by the source. It decreases, when particles are absorbed by solids and/or the background. Especially if a multipacting event is expected, this type of plot can be very useful.

Emitted Current (*NT: 1D Results* \Rightarrow *Emission Information* \Rightarrow *Current* \Rightarrow [*Sources*])



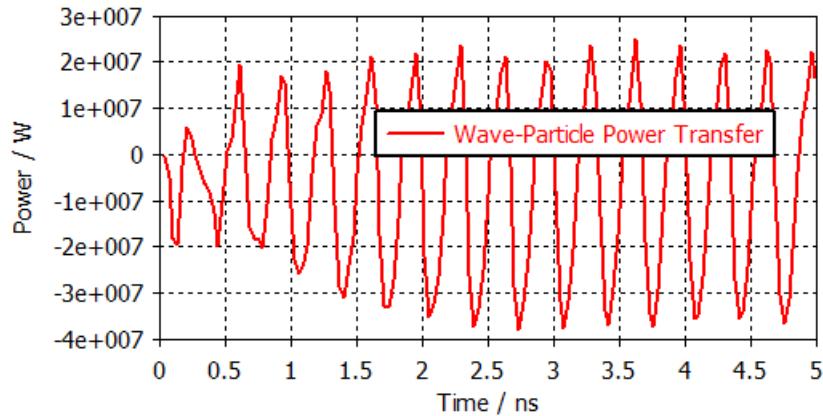
This 1D result shows the amount of emitted current for all particle sources vs. time. Especially for field based emission models, like explosive emission, this result is very important.

Collision Current (NT: 1D Results \Rightarrow Collision Information \Rightarrow Current \Rightarrow [Background])



The current that is absorbed by solids and/or the background due to collisions with particles is a standard 1D result of the PIC solver. A power plot is also available.

Wave-Particle-Power Transfer (NT: 1D Results \Rightarrow Power)



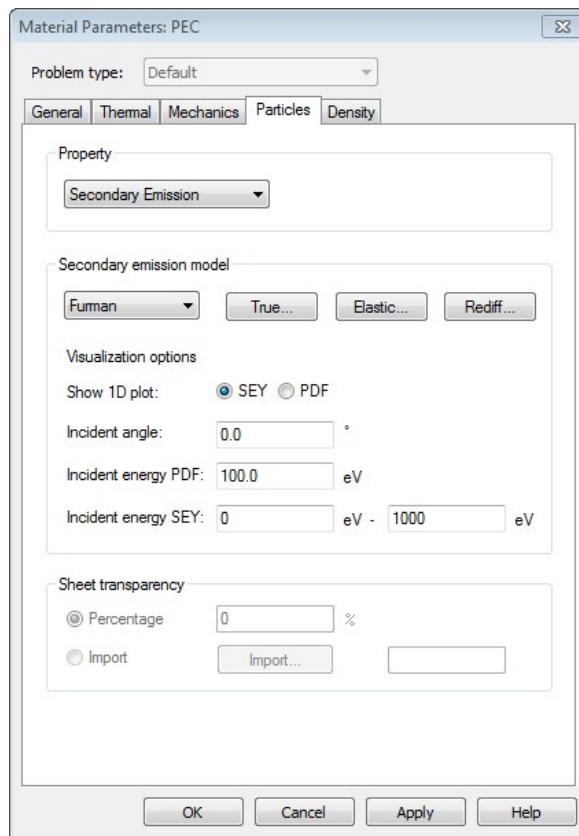
$$P_{\text{transfer}} = \int_V \vec{j} \cdot \vec{E} dv$$

The wave-particle power transfer is the power (loss or gain) that is transferred from the electromagnetic fields to the particles. In case of oscillators, this quantity can be very interesting. Superposed fields, i.e. analytic fields and field imports, are not taken into account for this plot.

There are even more possibilities for monitoring the particle data during the simulation and for analyzing the results, but the previously presented methods provide a good starting basis. For further options we would like to refer to the online help.

Additional information: Interaction of particles with materials

Particles can interact not only with electromagnetic fields but also directly with materials. To activate and edit the settings of the particle-material interaction, you can open the dialog box of a previously selected material with *Modeling: Materials* \Rightarrow *New/Edit*  \Rightarrow *Material properties* and click on the tab *Particles*. The following dialog box will then be visible.



Via the drop-down list in the *Property* frame, you can select the kind of particle-material interaction. Two properties are available: *Secondary Emission* (for electrons) and *Sheet Transparency*.

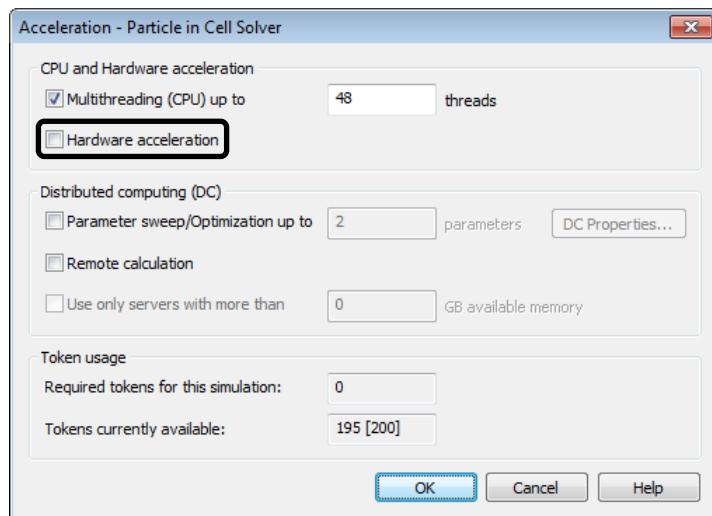
Secondary emission occurs when primary incident particles of sufficient energy hit a surface and induce the emission of secondary particles. In the frame *Secondary emission model*, the parameters of the secondary emission model can be specified. Options include a phenomenological probabilistic model (Furman), a heuristic model (Vaughan) and a model based on an imported secondary electron yield (Import).

In some applications very small grids or foils are present, where only part of the existing particles can pass. This can be represented by an infinitely thin body, so called sheet, which can become transparent to particles. In the frame *Sheet transparency*, the degree of this transparency can be specified, which can be either constant or energy-dependent.

Additional information: Acceleration Features

In addition to optimization and parameter sweep techniques, CST PARTICLE STUDIO offers several hardware-related possibilities to accelerate simulations. In order to specify the settings for the CPU and GPU acceleration and the distributed computing options within the PIC solver, select *Simulation* \Rightarrow *Solver* \Rightarrow *Start Simulation*  \Rightarrow *Acceleration*.

Please refer to the online help for more detailed information about the different acceleration features and the hardware that is needed. If you have an appropriate GPU try to enable the GPU acceleration feature and start the solver again. Currently the PIC solver supports one GPU.



Summary

This example should have given you an overview of the key concepts of CST PARTICLE STUDIO. You should now have a basic idea of how to do the following:

1. Model the structures by using the solid modeler
2. Specify the solver parameters, check the mesh and start the simulation
3. Define particle and field monitors
4. Visualize the particle distribution and use the PIC solver statistics

If you are familiar with all these topics, you have achieved a very good starting point for further improving your usage of the PIC solver inside CST PARTICLE STUDIO.

For more information on a particular topic, we recommend that you browse through the online help system which can be opened by pressing the F1 key or clicking on the Help button  in the upper-right corner. If you have any further questions or remarks, do not hesitate to contact your technical support team. We also strongly recommend that you participate in one of our special training classes held regularly at a location near you. Ask your support center for details.

Simulation Workflow: Wakefield

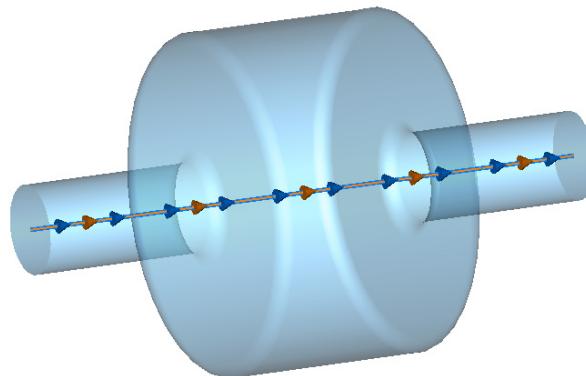
The following example demonstrates how to perform a wakefield calculation for a simple resonator cavity. Studying this example carefully will allow you to become familiar with many standard operations that are necessary to perform a wakefield simulation within CST PARTICLE STUDIO.

Go through the following explanations carefully even if you are not planning to use the software for wakefield simulations. Only a small portion of the example is specific to this particular application type since most of the considerations are general to all solvers and application domains.

The following explanations always describe the menu-based way to open a particular dialog box or to launch a command. Whenever available, the corresponding toolbar item is displayed next to the command description. Due to the limited space in this manual, the shortest way to activate a particular command (i.e. by pressing a shortcut key or activating the command from the context menu) is omitted. You should regularly open the context menu to check available commands for the currently active mode.

The Structure

This workflow example considers a particle beam passing through a pillbox cavity. Since only the vacuum parts of the structure need to be modeled, it is very easy to set up the geometrical description. It consists only of two added cylinders with a couple of blended edges. The following picture shows the structure of interest. It is shown in a transparent way, in order to see the particle beam axis.



CST PARTICLE STUDIO allows you to define the properties of the background material. Anything you do not fill with a particular material will automatically be considered as background material. For this structure, it is sufficient to model only the vacuum space. The background properties will be set to PEC (Perfect Electric Conductor).

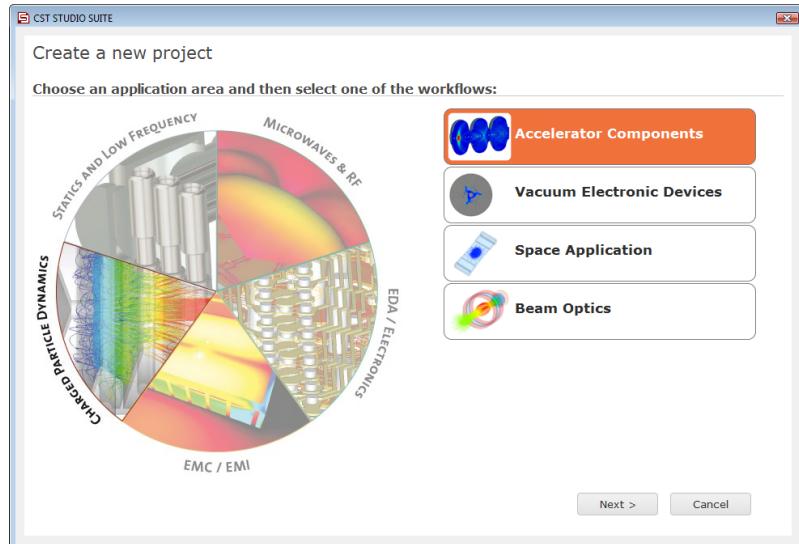
The model will be created in three simple steps:

1. Model the cylindrical vacuum parts of the resonator and the beam tube.
2. Blend the circular edges of the cavity.
3. Define the beam parameters (axis, charge, velocity).

Create a New Project

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

Next you should choose the application area, which is *Charged Particle Dynamics* for the example in this tutorial and then select the workflow by double-clicking on the corresponding entry.



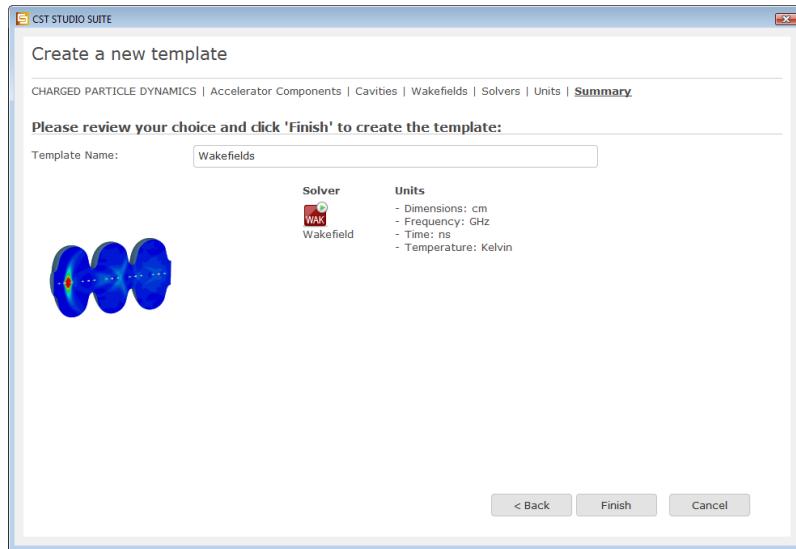
For the pillbox cavity, please select *Accelerator Components* \Rightarrow *Cavities* \Rightarrow *Wakefields* \Rightarrow *Wakefield* .

At last you are requested to select the units which fit your application best. For this example, please select the dimensions as follows:

Dimensions:	cm
Frequency:	GHz
Time:	ns

For the specific application in this tutorial the other settings can be left unchanged. After clicking the Next button, you can give the project template a name and review a summary of your initial settings:

Finally click the Finish button to save the project template and to create a new project with appropriate settings. CST PARTICLE STUDIO will be launched automatically due to the choice of this specific project template within the application area *Charged Particle Dynamics*.



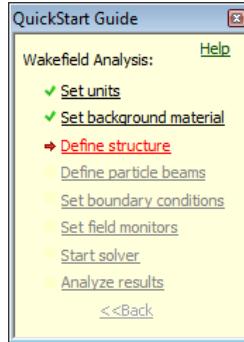
Please note: When you click again on the File: New and Recent you will see that the recently defined template appears below the Project Templates section. For further projects in the same application area you can simply click on this template entry to launch CST PARTICLE STUDIO with useful basic settings.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

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

Open the Wakefield QuickStart Guide

An interesting feature of the online help system is the QuickStart Guide, an electronic assistant that will guide you through your simulation. If it does not show up automatically, you can open this assistant by selecting QuickStart Guide from the Help button  in the upper right corner.

The following dialog box should then be visible at the upper right corner of the main view:



As the project template has automatically set the Solver type. Also Units and background settings have been predefined by the project template.

The red arrow always indicates the next step necessary for your problem definition. You may not have to process the steps in this order, but we recommend you follow this guide at the beginning in order to ensure all necessary steps have been completed.

Look at the dialog box as you follow the various steps in this example. You may close the assistant at any time. Even if you re-open the window later, it will always indicate the next required step.

If you are unsure of how to access a certain operation, click on the corresponding line. The *QuickStart Guide* will then either run an animation showing the location of the related Ribbon entry or open the corresponding help page.

Define the Units

The *Wakefields* template has already made some settings for you. The defaults for this structure type are geometrical units in cm and times in ns. You can change these settings by entering the desired settings in the units dialog box (*Home: Settings \Rightarrow Units* ) \Rightarrow Units), but for this example you should just leave the settings as specified by the template. Additionally, the used units are also displayed in the status bar:



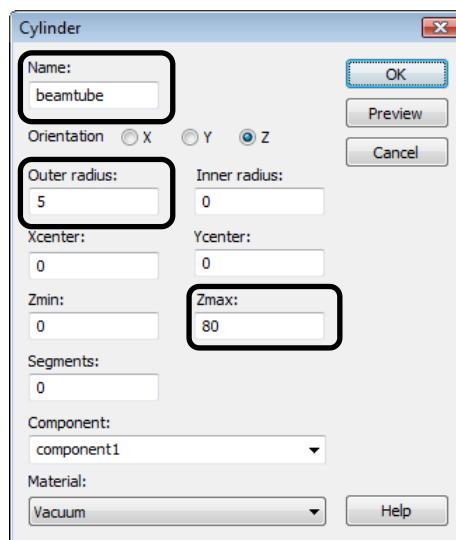
Define the Background Material

As previously discussed in the Structure section, the pillbox cavity is surrounded by perfect electrical conductor (PEC). The material type *PEC* is already set as default background material in the *Wakefields* template. You may change the background material in the corresponding dialog box *Simulation: Settings \Rightarrow Background*. For this example no change of the background material is needed.

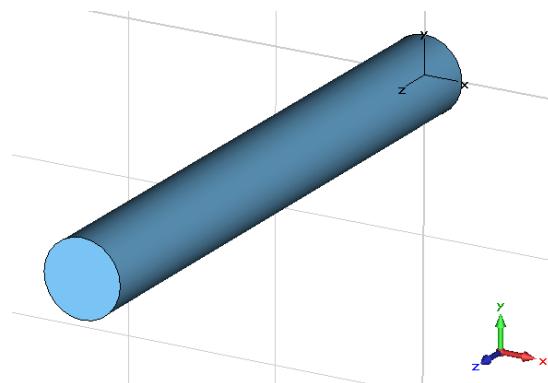
Model the Structure

First, create a cylinder along the z-axis of the coordinate system using the following steps:

1. Select the cylinder creation tool: *Modeling: Shapes \Rightarrow Cylinder*.
2. Press the *Shift+Tab* key, and enter the center point (0,0) in the xy-plane before pressing the *Return* key to store this setting.
3. Press the *Tab* key again, enter the outer-radius as 5 and press the *Return* key.
4. Press the *Tab* key, enter the height as 80 and press the *Return* key.
5. Press *Esc* to create a solid cylinder (skip the definition of the inner radius).
6. In the shape dialog box, enter “beamtube” in the *Name* field.

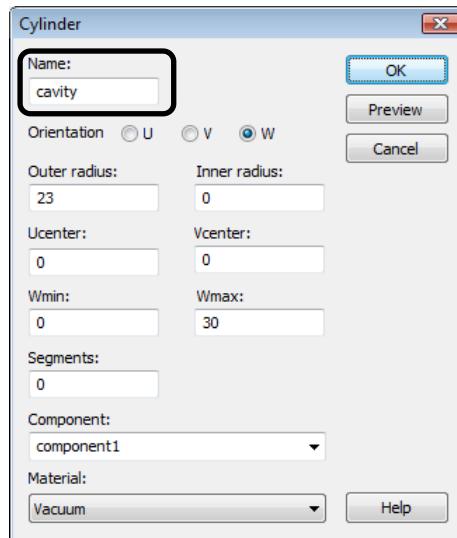


Since the material type “Vacuum” is already predefined, you can create the cylinder without defining a new material by clicking *OK*. Your result should look like the picture below. Press the *Space bar* to zoom the cylinder to window size.

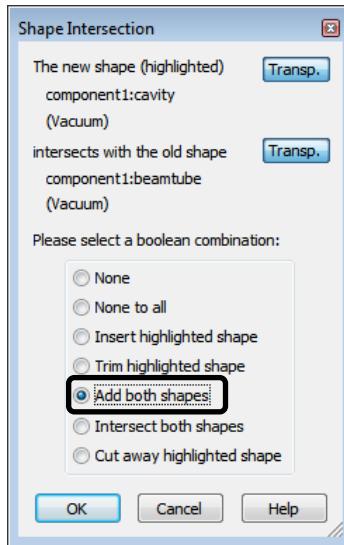


To create the cavity, you will now construct another vacuum cylinder with the help of the working coordinate system (WCS):

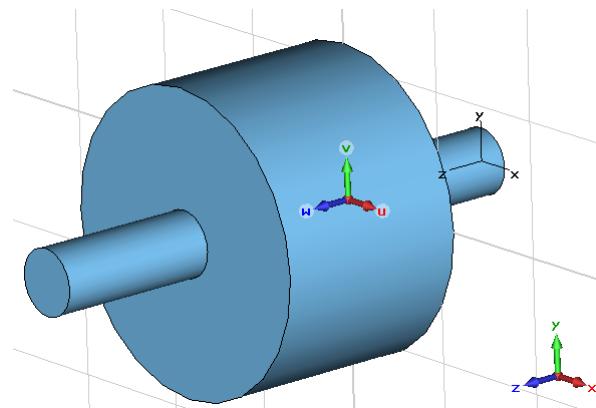
1. Activate the working coordinate system *Modeling: WCS* \Rightarrow *Local WCS*
2. Choose *Modeling: WCS* \Rightarrow *Transform WCS* , enter a shift of 25 in the *DW* direction and click on *OK*.
3. Again select the cylinder creation tool: *Modeling: Shapes* \Rightarrow *Cylinder* .
4. Press the *Shift+Tab* key, and enter the center point (0,0) in the *uv-plane* before pressing the *Return* key to store this setting.
5. Press the *Tab* key again, enter the outer-radius as 23 and press the *Return* key.
6. Press the *Tab* key, enter the height as 30 and press the *Return* key.
7. Press *Esc* to create a solid cylinder (skip the definition of the inner radius).
8. In the shape dialog box, enter “cavity” in the *Name* field.



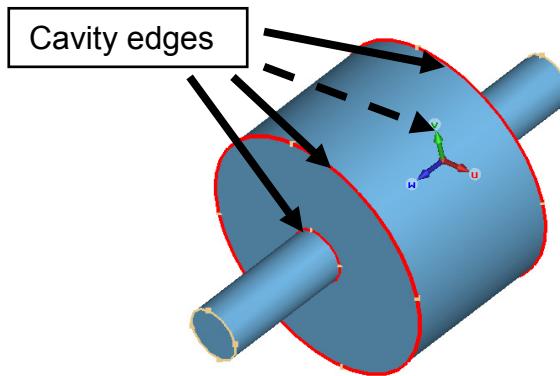
Confirm your setting by pressing *OK*. The automatic intersection check detects that both cylinders are intersecting and ask how to resolve the overlap:



It is important for the following construction steps to add both shapes to one. To this aim, select "Add both shapes" and confirm with *OK*.



The final construction step is to blend the outer circular edges at the cavity and the intersection edges between the cavity and the beam-tube. Since four edges have to be blended in one step you can activate the *Keep Pick Mode* tool *Modeling: Picks* \Rightarrow *Picks* \Rightarrow *Pick Modes* \Rightarrow *Keep Pick Mode* before picking the four edges. Now activate *Modeling: Picks* \Rightarrow *Pick Points, Edges or Faces* to pick the first edge – you might also use the keyboard shortcut e:

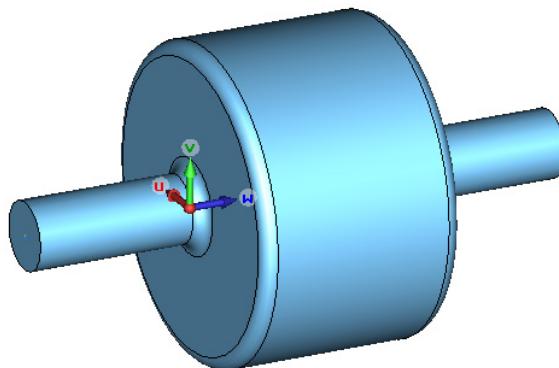


By moving the mouse cursor to the first edge and performing a double-click you select the appropriate edge. Repeat this operation for the other three circular edges on the cylindrical cavity.

Now press the *Return* key to store all picks. Deactivate *Modeling: Picks* \Rightarrow *Picks* \Rightarrow *Pick Modes* \Rightarrow *Keep Pick Mode* by selecting it once more. To activate the blend tool finally select *Modeling: Tools* \Rightarrow *Blend* \Rightarrow *Blend Edges* and enter the value 2 for the blend radius.



Confirm with OK. Now the work of defining the geometric part is done, and your model should look as follows (after switching off the visualization of the working plane by pressing the *Alt+W* keys):

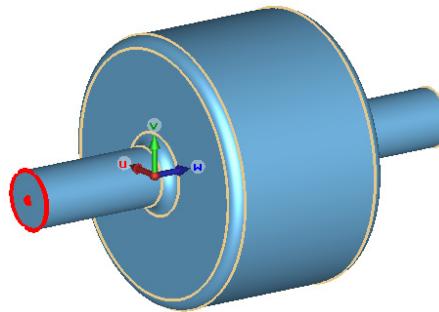


Congratulations! You have just created your first wakefield structure within CST PARTICLE STUDIO.

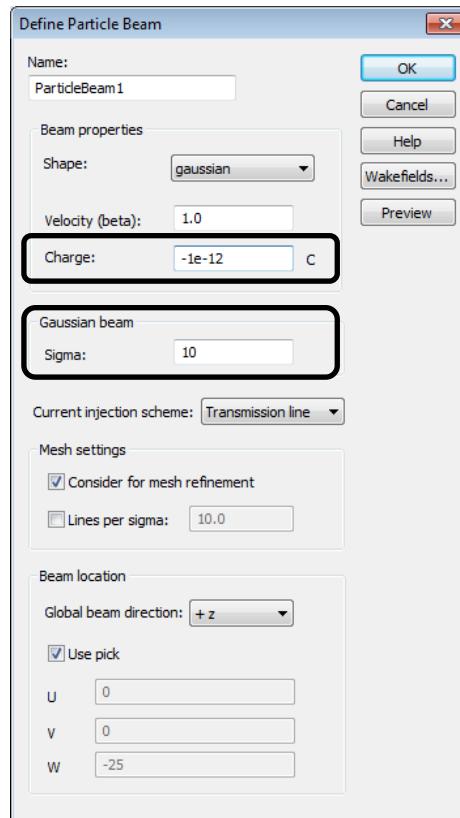
Define the Particle Beam Source

A wakefield computation is always driven by a particle beam source, which will be defined in this section. The beam definition consists of the axis settings and the description of a charged bunch of particles with a Gaussian shape.

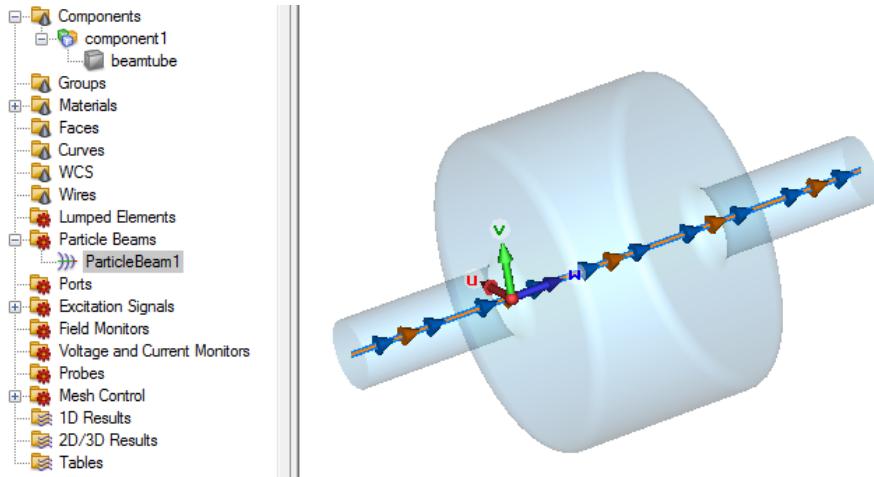
1. Select *Modeling: Picks* \Rightarrow *Pick Point* \Rightarrow *Pick Circle Center*
2. Double-click on the lower circular edge of the beam tube with respect to the z axis. The selected edge will be highlighted:



3. Open the particle beam dialog box by selecting *Simulation* \Rightarrow *Sources and Loads* \Rightarrow *Particle Beam*



Enter a value of 10 (cm) for the longitudinal spatial width of the Gaussian pulse, and a total bunch charge of -1e-12 C. Confirm the settings with the *OK* button and the beam source is created. Since the structure is hiding the source visualization you might select *NT: Particle Beams* to take a look at the beam source:

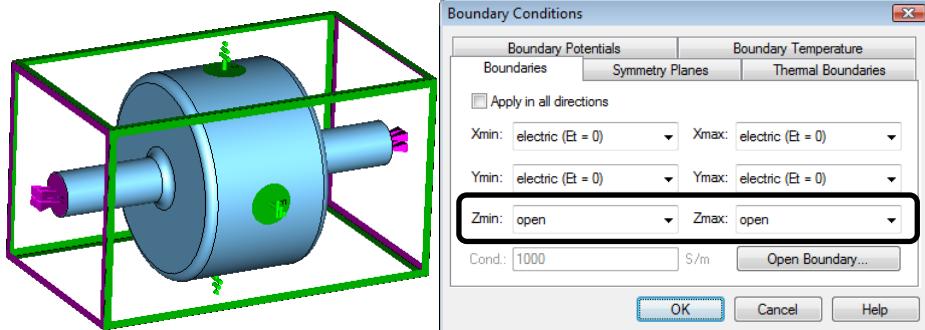


Note: The blue arrows indicate the beam position, whereas the orange arrows indicate the position of the wake integration path.

Define Boundary and Symmetry Conditions

The simulation of this structure will only be performed within the bounding box of the structure. You may, however, specify certain boundary conditions for each plane ($X_{min}/X_{max}/Y_{min}/Y_{max}/Z_{min}/Z_{max}$) of the bounding box.

The boundary conditions are specified in a dialog box which opens after choosing *Simulation: Settings \Rightarrow Boundaries*.

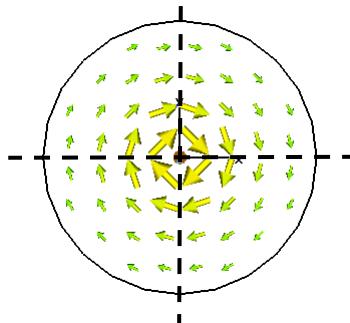


While the boundary dialog box is open, the boundary conditions will be visualized in the structure view as in the picture above.

In this simple case, the structure is embedded in perfect conducting material, so all x- and y- boundary planes may be specified as “electric” planes (which is the default). The z-boundaries are defined as “open” planes, such that eventual scattering fields traveling along the beam tube can be absorbed at the lower and upper z-boundaries.

In addition to these boundary planes, you can also specify "symmetry planes." The specification of each symmetry plane will reduce the simulation time by a factor two.

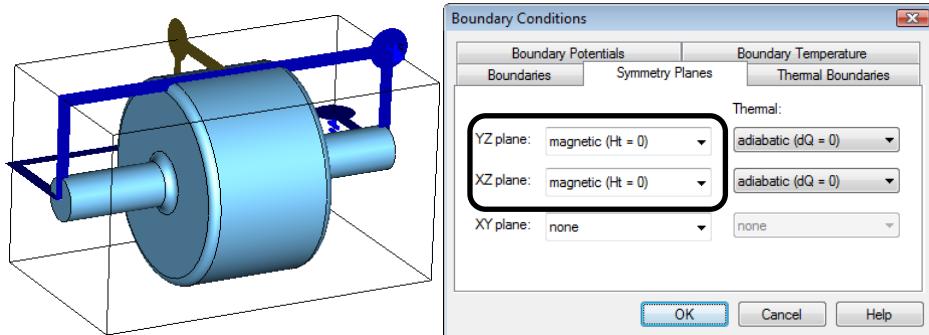
In our example, the structure is rotationally symmetric with respect to z-axis, therefore the yz-plane and the xz-plane can be set to be symmetry planes. The excitation of the fields will be performed by the particle beam source for which the magnetic field is shown below:



Plane of structure symmetries (yz- and xz-planes) illustrated by means of the magnetic field.

The magnetic field has no component tangential to the planes of the structure's symmetry (the entire field is oriented perpendicular to this plane). If you specify these planes as "magnetic" symmetry planes, you can direct CST PARTICLE STUDIO to limit the simulation to one quarter of the actual structure while taking the symmetry conditions into account.

For the YZ and XZ symmetry planes, you can choose *magnetic* either by selecting the appropriate option in the dialog box or by double-clicking on the corresponding symmetry plane visualization in the view and selecting the proper choice from the context menu. Once you have done so, your screen will appear as follows:



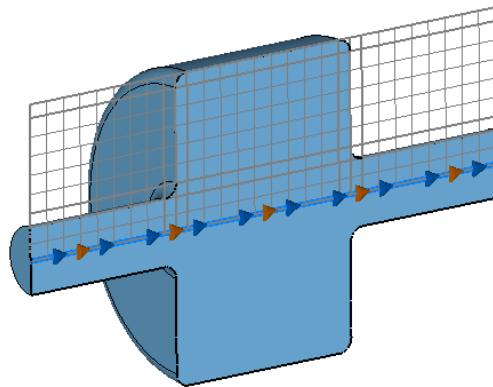
Symmetry Planes tab in the boundary conditions dialog box.

Finally click *OK* in the dialog box to store the settings. Then the boundary visualization will disappear.

Visualize the Mesh

The mesh generation (hexahedral mesh) for the structure's analysis is performed automatically based on an expert system. However, in some situations it may be helpful to inspect the mesh to improve the simulation speed by changing the parameters for the mesh generation.

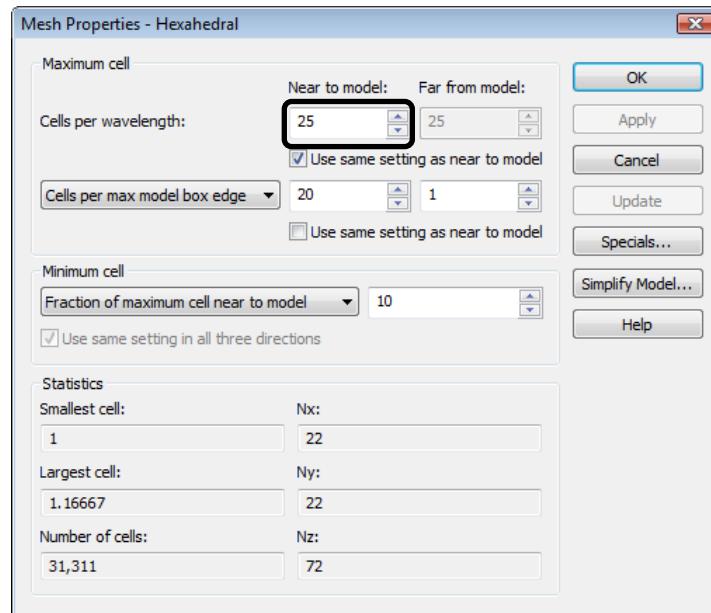
The mesh can be visualized by entering the mesh view *Home: Mesh* \Rightarrow *Mesh View* . For this structure, the mesh information will be displayed as follows:



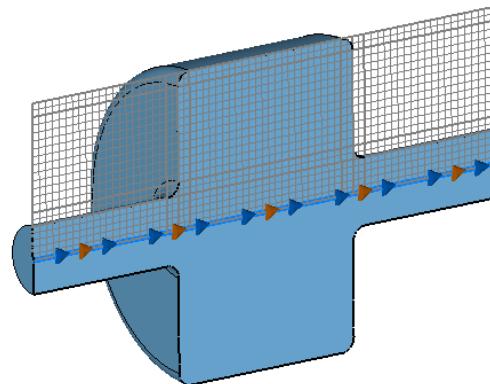
One 2D mesh plane is in view at a time. Because of the symmetry setting, the mesh plane extends across only one half of the structure. You can modify the orientation of the mesh plane by adjusting the selection in the *Mesh: Sectional View* \Rightarrow *Normal* dropdown list or just by pressing the X/Y/Z keys. Move the plane along its normal direction using the Up/Down cursor keys. The current position of the plane will be shown in the *Mesh: Sectional View* \Rightarrow *Position* field.

There are some thick mesh lines shown in the mesh view. These mesh lines represent important planes (so-called snapping planes) at which the expert system finds it necessary to place mesh lines. You can control these snapping planes in the Special Mesh Properties dialog by selecting *Simulation: Mesh* \Rightarrow *Global Properties*  \Rightarrow *Specials* \Rightarrow *Snapping*.

For wakefield computations the minimization of dispersion due to the mesh is very important, especially in longitudinal beam direction. Therefore the particle bunch has to be sampled adequately in space. Open the mesh properties dialog box by selecting *Home: Mesh \Rightarrow Global Mesh Properties*.



This example is driven by quite a long bunch (compared to the structure's dimensions), therefore the sampling rate can be increased by entering a value of 25 for the *Lines per wavelength* setting. In case the bunch length is very short, this might increase the number of mesh cells drastically. However, a simulation is still possible using cluster simulation via MPI. Please refer to the Online Help->Simulation Acceleration -> MPI Computing. Leave the dialog box by clicking *OK* and have a look at the refined mesh:



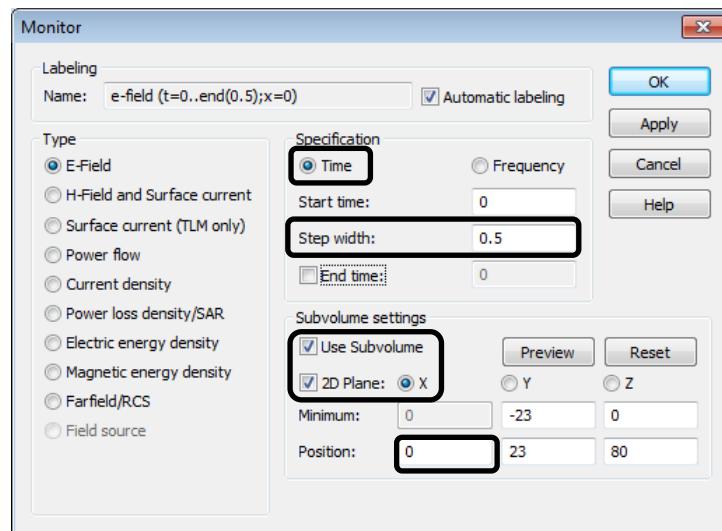
You should now leave the mesh inspection mode by toggling *Mesh: Close \Rightarrow Close Mesh View*.

Define a 2D Time Domain Field Monitor

To understand the behavior of an electromagnetic device, it is often useful to get insight into the electromagnetic field distribution. In this example, it may be interesting to see where electric fields are created by the particle bunch.

The fields can be recorded at arbitrary frequencies or with a given sampling rate in the time domain. Since storing all computed field data would require a tremendous amount of memory space only samples are stored. To obtain these field samples so called monitors have to be defined.

Monitors can be defined in a dialog box that opens after choosing *Simulation: Monitors* \Rightarrow *Field Monitor* . You may need to switch back to the modeler mode by selecting the *Components* folder in the navigation tree before the monitor definition is activated.



After selecting the proper *Type* for the monitor, you may specify its time settings in the *Specification* field. Clicking *Apply* stores the monitor while leaving the dialog box open. All time settings are specified in the active time unit, which was previously set to “ns”. For this analysis you should enter the following settings:

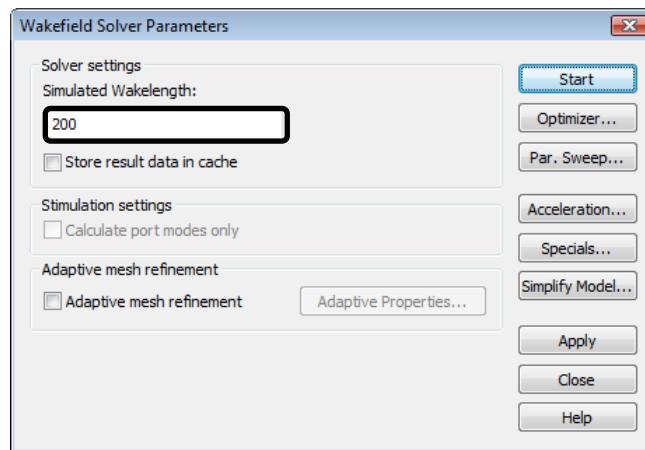
Field type	E-Field
Specification	Time
Start time	0
Step width	0.5
2D Plane	Activate on
Orientation	X
Position	0

Finally leave the dialog box by clicking *OK*. All defined monitors are listed in the *NT: Monitors* folder. Within this folder you may select a particular monitor to reveal its parameters in the main view.

Note: After the simulation has been finished, you can visualize the recorded field by choosing the corresponding item from the navigation tree. The monitor results can be found then in the *NT: 2D/3D Results* folder. The results are ordered according to their physical quantity *E-Field / H-Field / Currents / Power flow*.

Start the Simulation

After having defined all necessary parameters, you are ready to start the wakefield simulation. Start the simulation from the Wakefield Solver control dialog box: *Simulation: Solver* \Rightarrow *Setup Solver* .



In this dialog box, you can specify the maximum wakelength behind the bunch which should be calculated. Enter a value of 200 (cm) in this field.

The accuracy of the results mainly depends on the discretization of the structure and can be improved by refining the mesh. In case a resonant structure is observed, a short simulated wakelength introduces a truncation error in the wake potential. This could lead to ripples in the wake impedance.

You can now start the simulation procedure by clicking the *Start* button. A progress bar will appear in the status bar which will inform you on the solver's progress. Information text regarding the operation will appear next to the progress bar. The most important stages are listed below:

1. Calculating matrices, preparing and checking model:

During this step, your input model is checked for errors such as invalid or overlapping materials.

2. Calculating matrices, normal matrix and dual matrix:

During these steps, the system of equations, which will subsequently be solved, is set up.

3. Transient analysis, calculating the port modes:

In this step, the solver calculates the port mode field distributions if any ports were defined. This information will be used later in the time domain analysis of the structure.

4. Transient analysis, processing excitation:

During this stage, the particle beam is injected into the calculation domain. The solver then calculates the resulting field distribution inside the structure as well as the wakefields.

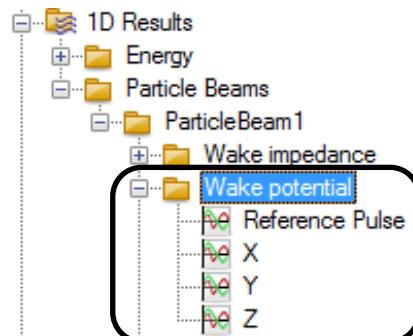
5. Transient analysis, transient field analysis:

After the beam pulse has been injected, the solver continues to calculate the field distribution and the wake potentials until the requested wakelength has been computed.

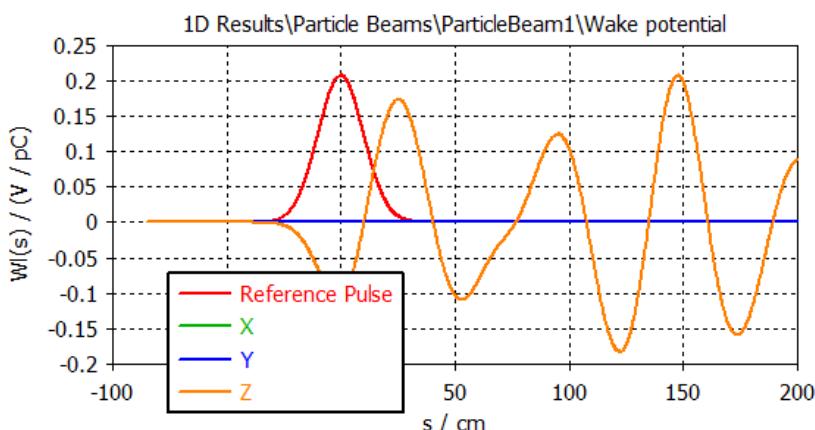
For this simple structure, the entire analysis takes only a few seconds to complete.

Analyze the Simulation Results

After the solver has completed the wake computation, you can view the results. In order to look at the wake potential, choose the solution from the navigation tree. You can visualize them by selecting *NT: 1D Results* \Rightarrow *Particle Beams* \Rightarrow *ParticleBeam1* \Rightarrow *Wake potential*. If you open this subfolder, you will see all signals assigned to that folder.

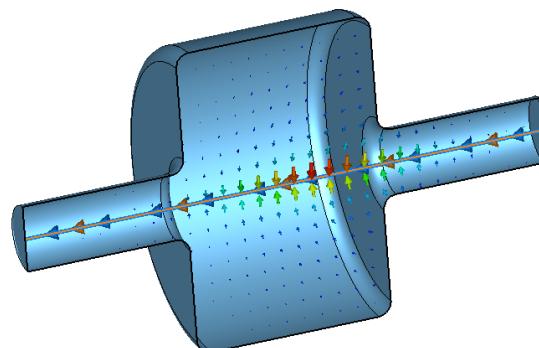


After selecting the folder you should see the following plot:

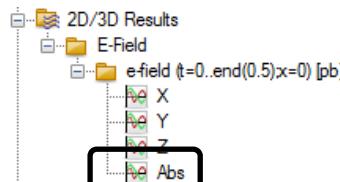


The *Reference Pulse* graph is shown only for orientation purposes. As expected due to the symmetry of structure and beam, only the longitudinal z-wake potential is different from zero.

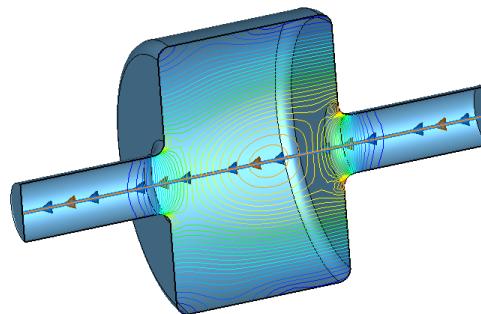
If you select the electric field result from the previously defined monitor *NT: 2D/3D Results* \Rightarrow *E-Field* \Rightarrow *e-field (...) [pb]*, you may obtain a plot showing no arrows at all. This is due to the fact that the first time sample has been selected automatically at a time where the beam has not yet entered the calculation domain. Deactivate the all transparent mode *2D/3D Plot: View Options* \Rightarrow *All Transparent*  and select another time frame by using the left / right cursor keys when the focus is in the main window.



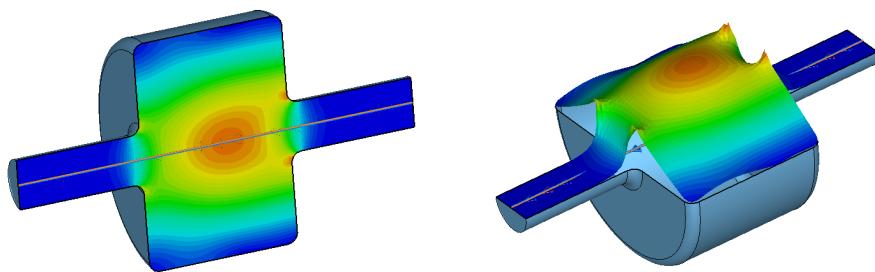
Not all plot options and modifications can be explained here. Please refer to the Online Help for more details. However, select the absolute value of the electric field *NT: 2D/3D Results* \Rightarrow *E-field* \Rightarrow *e-field (...)* \Rightarrow *Abs*.



The following gallery shows some possible plot options for the absolute electric field values. Can you reproduce them?



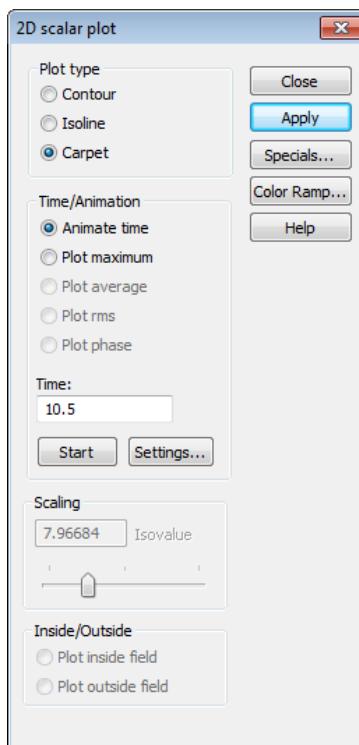
Isoline plot of the absolute E-field



Contour plot of the absolute E-field

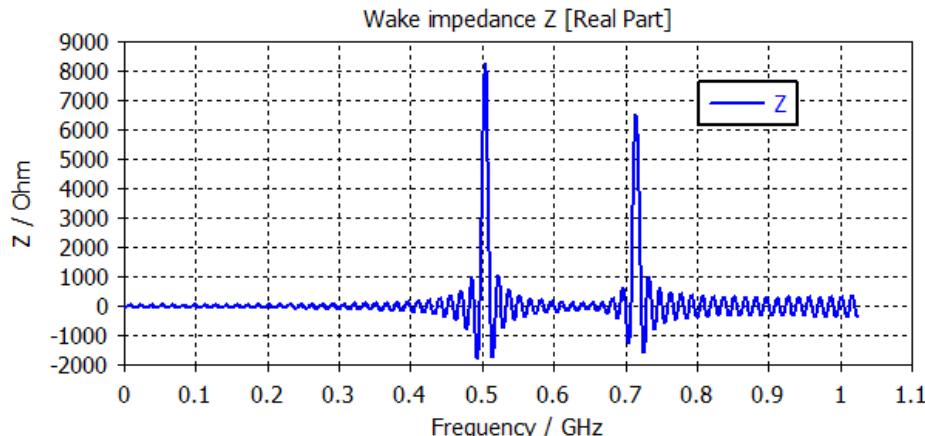
Carpet plot of the absolute E-Field

Hint: To reproduce the pictures above, open the *2D Scalar Plot* dialog box *2D/3D Plot: Plot Properties* :



Additional Information: Wake Postprocessor

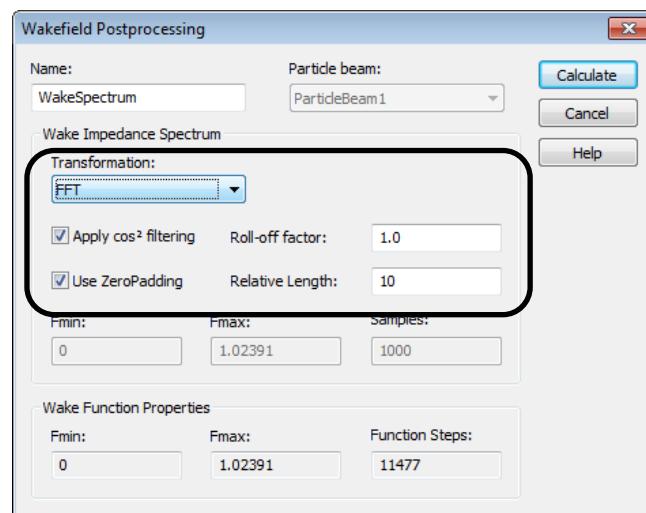
During the solver run complex wake impedances are computed by dividing the wake potential by the charge distribution of the beam in frequency domain. These impedances are accessible from the navigation tree. The following picture shows the real part of the Z-impedance for the previous example with a *Simulated Wakelength* setting of **2000**:



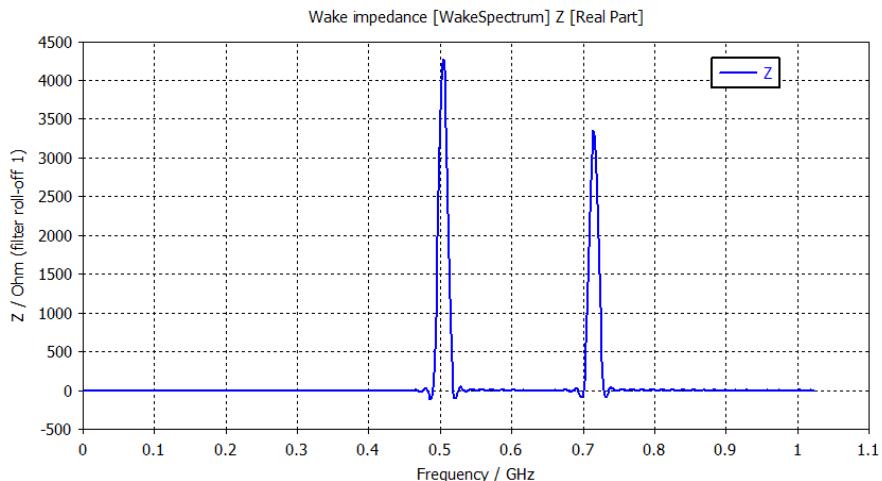
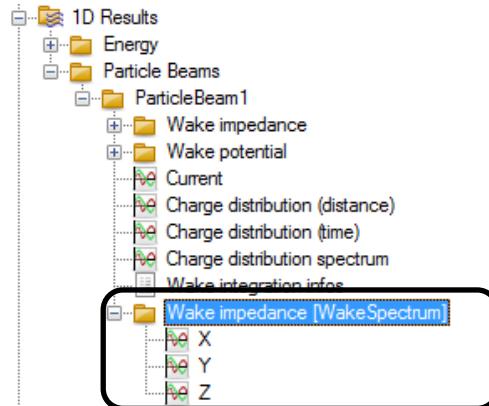
Real part of the z-wake impedance for the previous example.

This impedance shows the typical truncation error (ripples) for a time signal which has not decayed to zero before the simulation was completed. In this particular case, the wake potential is truncated in time domain.

It is possible to recalculate the impedance spectra after a simulation has been finished by selecting *Post Processing: 2D/3D Field Post Processing* \Rightarrow *Wakefield Postprocessing*:



This post processing option allows recomputing of the wake impedances. Additionally, a low-pass filter can be applied to the impedance to smoothen the signal. Moreover it is possible to recompute certain frequency intervals with a given sampling rate (only for DFT transformation type). For a very fast computation of the complete spectrum use the FFT transformation type. The impedance spectra can be accessed by selecting *NT: 1D Results* \Rightarrow *Particle Beams* \Rightarrow *ParticleBeam1* \Rightarrow *Wake impedance [Name]* \Rightarrow *Z*:



Real part of the z- wake impedance computed with a \cos^2 - filter and the FFT transformation type.

The wake impedance describes the behavior of the cavity in the frequency domain. For this type of impedance the beam serves as a current source and the wake potential as voltage. Thus this impedance can be used to detect the modes where beam and structure interact.

Note: The DFT transformation type is helpful when computing only a few samples within a specified frequency range, while the FFT type computes a full spectrum very fast.

Summary

This example should have given you an overview of the key concepts of CST PARTICLE STUDIO. You should now have a basic idea of how to do the following:

1. Model the structures by using the solid modeler
2. Specify the solver parameters, check the mesh and start the simulation
3. Visualize the wake potentials and impedance profiles
4. Define field monitors
5. Visualize the electromagnetic field distributions

If you are familiar with all these topics, you have a very good starting point for further improving your usage of CST PARTICLE STUDIO.

For more information on a particular topic, we recommend that you browse through the online help system which can be opened by selecting *File: Help ⇔ CST STUDIO SUITE – Help*. If you have any further questions or remarks, please do not hesitate to contact your technical support team. We also strongly recommend that you participate in one of our special training classes held regularly at a location near you. Please ask your support center for details.

Chapter 3 – Coupled Simulations

CST PARTICLE STUDIO offers various options to link electromagnetic field simulations to a specific particle computation. Furthermore the Particle Interfaces allow linkage of different tracking or PIC simulations. Finally one can export losses from crashed particles to a subsequent thermal analysis. Usually it is either possible to perform several simulations within a single project or connect two or more projects by using the import and export options.

Considering Electromagnetic Fields

CST PARTICLE STUDIO is dedicated to simulate charged particles traveling through electromagnetic fields. To accomplish this task, one (or more) of three possible techniques can be used:

1. Computation of electromagnetic fields
2. Definition of analytic magnetic fields
3. Import of electromagnetic fields - ASCII or from other projects

In general, all fields defined for a PIC or tracking simulation are superposed before being used for the particle update. Specifically in case of the PIC solver, these fields are superposed to the self-consistent and time-dependent fields based on Maxwell's equations.

Computation of Electromagnetic Fields

CST PARTICLE STUDIO has the ability to use fields from other CST STUDIO SUITE 3D EM solvers as input, particularly CST EM STUDIO and CST MICROWAVE STUDIO.

- **Electrostatics Solver** 

The Electrostatics Solver of CST EM STUDIO is used to calculate the accelerating fields for static guns, or the deflecting electrostatic fields of beam steering units in cathode ray tubes (CRT).

- **Magnetostatics Solver** 

CST EM STUDIO's Magnetostatics Solver pre-calculates the fields of various types of magnets (such as solenoids, dipoles, quadrupoles, etc.) for beam optics simulation.

- **Eigenmode Solver** 

The particles can also be tracked through resonant fields in cavities calculated with CST MICROWAVE STUDIO's Eigenmode Solver.

- **Time Domain Solver** 

The particles can also be tracked through the frequency domain 3D field monitors of CST MICROWAVE STUDIO's Time Domain Solver. A typical application is multipaction analysis.

To get an introduction and/or further information to these electromagnetic field solvers, refer to the *Workflow and Solver Overview* of CST EM STUDIO and CST MICROWAVE STUDIO.

Definition of Analytic Magnetic Fields

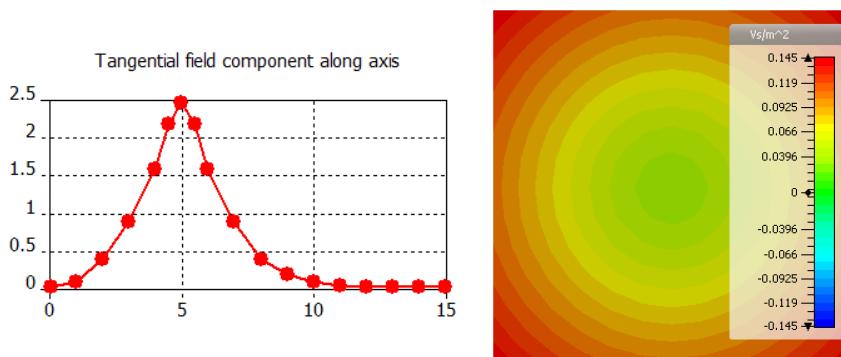
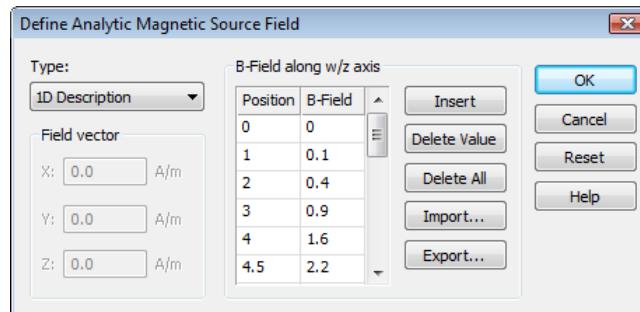
Besides the possibility of calculating fields before or during a particle simulation, CST PARTICLE STUDIO offers the option to define and use analytical H- and B-field distributions for the Tracking- and the PIC-solver.

Three different types of analytic magnetic field distributions are currently available:

- A constant magnetic field throughout the computational domain
- A constant magnetic flux density throughout the computational domain
- A rotationally symmetric magnetic field characterized by a 1D tangential magnetization vector defined along the Z-/ W- axis of the active global or local coordinate system. The r-component of the rotationally symmetric magnetic field can only be calculated if the magnetic field's z-component is not a function of the radius r:

$$B_r(r, z) = -\frac{r}{2} \frac{\partial B_z(z)}{\partial z}$$

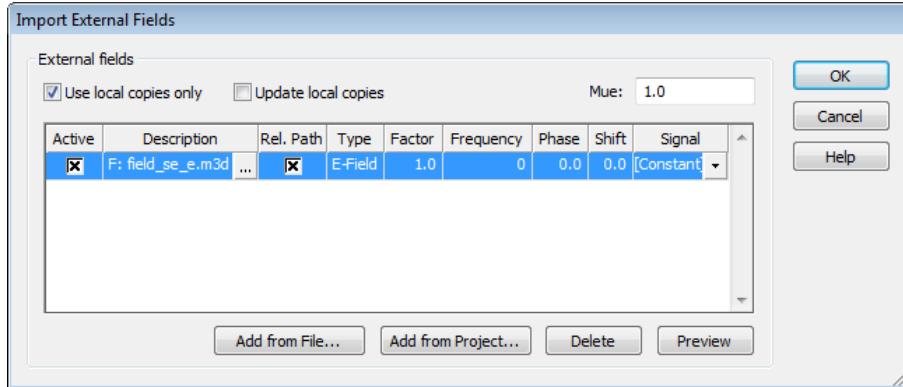
It is possible to define such a source by selecting *Simulation: Sources and Loads* \Rightarrow *Source field* \Rightarrow *Analytic Source Field* . The corresponding dialog box allows you to define the magnetic field vector. Alternatively, a 1D description of the magnetic field along the axis of the currently active coordinate system can be defined:



The picture above shows the “measured” tangential field along the z-axis and the rotationally symmetric field distribution of the resulting B-field.

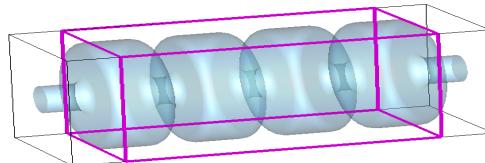
Import of Electromagnetic Fields

The third possibility to consider fields for a tracking or PIC simulation is to import them from an ASCII file or from another CST-project. Thus it is easily possible to superpose multiple fields. To define one or more field imports, open the dialog box by selecting *Simulation: Sources and Loads \Rightarrow Source Field \Rightarrow Import External Field*:

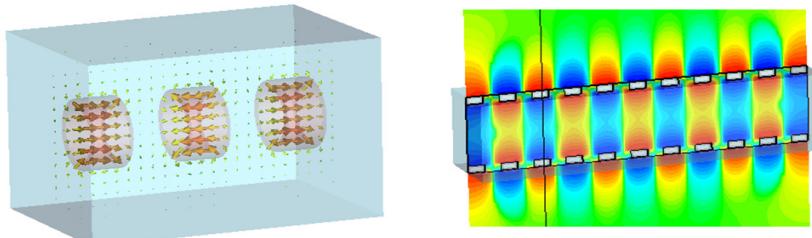


This feature allows importing of eigenmodes, e-, h- or b-fields even from different projects based on different meshes. When creating a field import with the *Add from Project* option, one can pick an existing field distribution from a CST project file fields based on hexahedral (HEX) and/or tetrahedral (TET) meshes can be imported. *Add from File* offers the possibility to import ASCII files or HEX mesh based monitor files.

By clicking the Preview button the overlapping regions of the imported data and the current domain can be visualized with a magenta colored frame.



It is possible to combine fields from different structures with a particle simulation, but care has to be taken since the program does not check the consistency of fields on material boundaries.



Another nice aspect is that a recalculation of tracking or PIC problems does not require the recalculation of fields. This results in a simulation speed up.

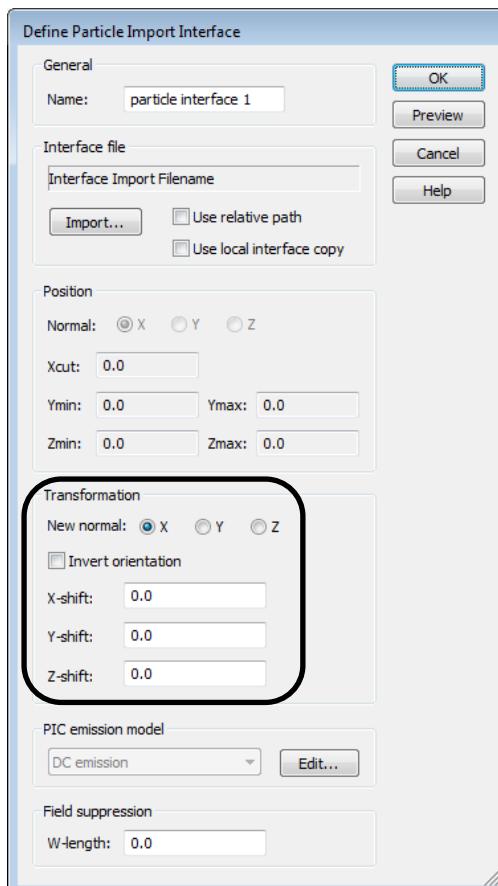
Particle Interfaces

Particle interfaces allow you to connect tracking and/or PIC simulations from different CST PARTICLE STUDIO projects. Two types of interfaces are available:

- Export Interface ➔
- Import Interface ➕

Assuming that you have a tracking or gun project which has to be linked to a subsequent PIC or tracking project by using Particle interfaces, perform the following steps to define a proper connection:

1. Open the tracking or gun project and run a tracking or gun simulation.
2. Define one or more export interfaces: *Simulation: Monitors* ➔ *Particle Monitor* ➔ *Particle Export Interface* ➔.
3. After a tracking or gun simulation, the particle data are automatically exported into a file with the extension .pio. This file is stored in the result folder of the project.
4. Open the PIC or tracking project.
5. Define one or more import interfaces by importing the particle interface files: *Simulation: Sources and Loads* ➔ *Particle Sources* ➔ *Particle Import Interface* ➕. It is possible to rotate and translate the interface plane.

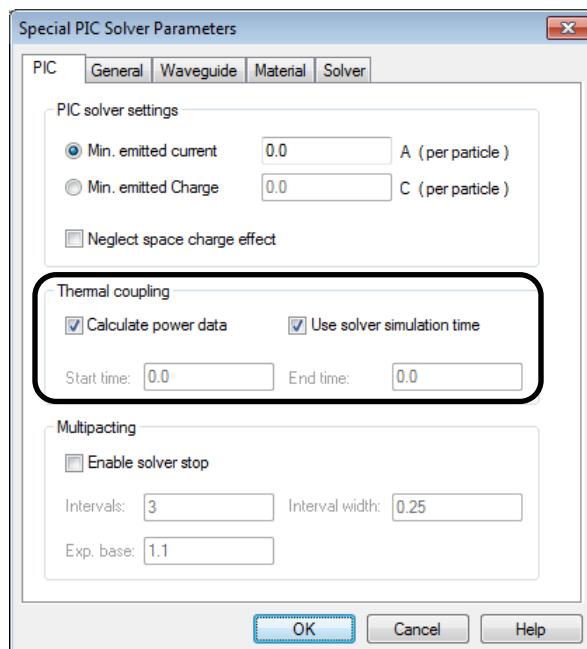


6. Run the subsequent PIC or tracking simulation.

Note: An ASCII import of files with *user defined* particle emission information is also available. Further information about the file format can be obtained from the online help.

Export of Thermal Losses

The PIC solver allows exporting thermal losses caused by particles interacting with matter. For example this might be an interesting option for medical applications, but also for collectors. It can be activated by opening *Simulation: Solver* \Rightarrow *PIC Solver* \Rightarrow *Specials* \Rightarrow *PIC*:



Since an averaged power is needed for the thermal coupling, the time period in that the power data are averaged has to be defined. Per default this time period is set to the user specified simulation time.

Of course, it is also possible to export thermal losses caused by electromagnetic fields. This is possibly an interesting option for wakefield or PIC computations. For further information about thermal coupling we refer to the CST MPHYSICS STUDIO help.

Chapter 4 — Find Further Information

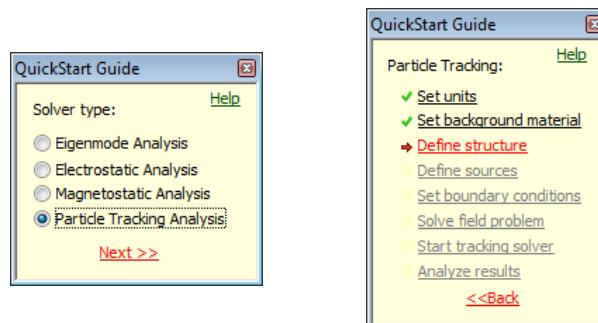
After carefully reading this manual, you will already have some idea of how to use CST PARTICLE STUDIO efficiently for your own problems. However, when you are creating your own first models, a lot of questions will arise. In this chapter we give you a short overview of the available documentation.

The QuickStart Guide

The main task of the QuickStart Guide is to remind you to complete all necessary steps in order to perform a simulation successfully. Especially for new users – or for those rarely using the software – it may be helpful to have some assistance.

The QuickStart Guide is opened automatically on each project start, when the checkbox *File: Options → Preferences → Open QuickStart Guide* is checked. Alternatively, you may start this assistant at any time by selecting *QuickStart Guide* from the Help button  in the upper right corner.

When the QuickStart Guide is launched, a dialog box opens showing a list of tasks, where each item represents a step in the model definition and simulation process. Usually, a project template will already set the problem type and initialize some basic settings like units and background properties. Otherwise, the QuickStart Guide will first open a dialog box in which you can specify the type of calculation you wish to analyze and proceed with the *Next* button:



As soon as you have successfully completed a step, the corresponding item will be checked and the next necessary step will be highlighted. You may, however, change any of your previous settings throughout the procedure.

In order to access information about the QuickStart Guide itself, click the *Help* button. To obtain more information about a particular operation, click on the appropriate item in the QuickStart Guide.

Online Documentation

The online help system is your primary source of information. You can access the help system's overview page at any time by choosing *File: Help*  *CST STUDIO SUITE – Help*  . The online help system includes a powerful full text search engine.

In each of the dialog boxes, there is a specific *Help* button which directly opens the corresponding manual page. Additionally, the *F1* key gives some context sensitive help when a particular mode is active. For instance, by pressing the *F1* key while a basic shape generation mode is active, you can get information about the definition of shapes and possible actions.

When no specific information is available, pressing the *F1* key will open an overview page from which you may navigate through the help system.

Please refer to the *CST STUDIO SUITE - Getting Started* manual to find some more detailed explanations about the usage of the CST PARTICLE STUDIO Online Documentation.

Examples

The installation directory of CST STUDIO SUITE contains an examples subdirectory consisting of a couple of typical application examples. A quick overview of the existing examples can be obtained by following the *Examples Overview* link on the online help system's start page.

Each of these examples also contains a “Readme” item in the navigation tree. By double-clicking on these items, you will obtain some information about the particular example regarding structure modeling and simulation procedure.

Although these examples are not explained in as much detail as the tutorials, they may nevertheless contain helpful hints which can be transferred to your particular application.

Technical Support

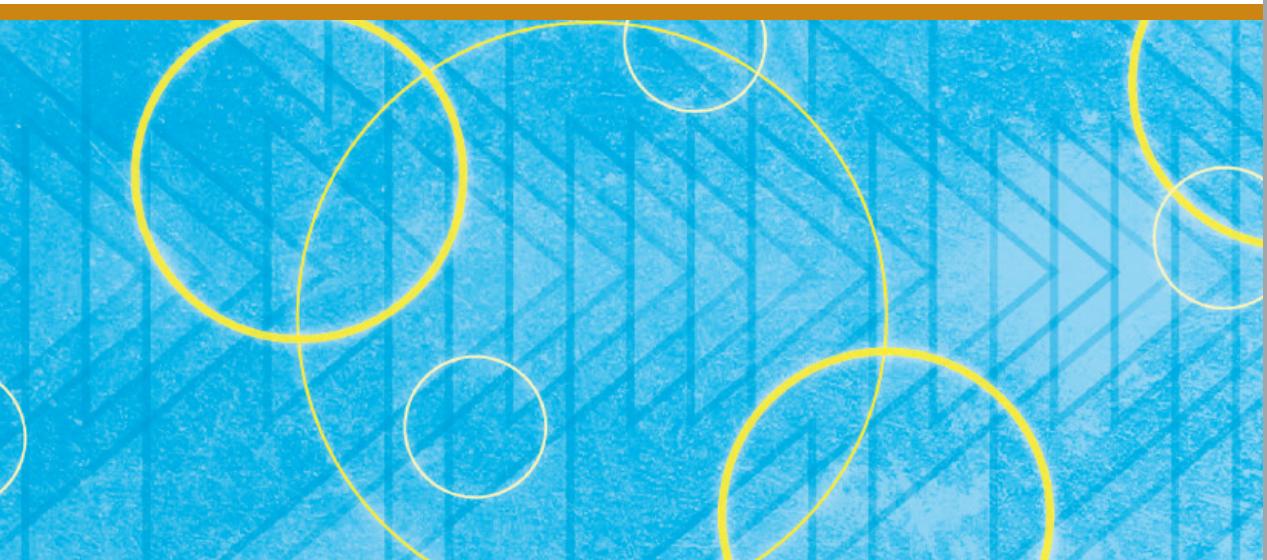
After you have taken your first steps to solving your own applications within CST MICROWAVE STUDIO, please use the *File: Project*  *Archive As* function to create an archive containing all relevant files. This archive should then be sent to the technical support team. Even if you have successfully obtained a solution, the problem specification might still be improved in order to get even better results within shorter calculation times.

The preferred option to contact technical support is to submit a so-called support ticket. You can create a new ticket or manage existing tickets from within the support area on our homepage or by selecting *File: Help*  *Support Tickets*.

The support area on our homepage (www.cst.com) also contains a lot of very useful and frequently updated information. Simple access to this area is provided by choosing *File: Help*  *Online Support Area*. You only need to enter your user name and password once. Afterwards, the support area will open automatically whenever you choose this command. Please note that the online help system's search function also allows searching in the online content as well.

History of Changes

An overview of all new main features of the release can be obtained by selecting the *Spotlight CST STUDIO SUITE 2015* page from the online help system (*File: Help ⇔ CST STUDIO SUITE – Help* ). A detailed *History of Changes* can also be accessed through the *Spotlight* page in the Online Help. The *Changes in the Service Packs* page in the same location in addition describes smaller changes released during intermediate service packs. Since there are many new features in each new version, you should browse through these lists even if you are already familiar with one of the previous releases.



© CST 2015 | CST – Computer Simulation Technology AG | www.cst.com

