

[Low-Frequency Guide](#) |

Chapter 12. Electric Field Analysis

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

Electric field analyses calculate the electric field in conductive or capacitive systems. Typical quantities of interest in an electric field analysis include:

- Electric field
- Current density
- Electric flux density
- Charge density
- Joule heat

This chapter describes elements used in all types of electric field analysis. You can use them to model electric effects in lossy dielectrics, high-voltage insulators, microwave passive components, semiconductor devices, micro-electromechanical (MEMS) devices, and biological tissues. This chapter also specifically covers the procedures for performing steady-state current conduction analysis and quasistatic time-harmonic and time-transient electric field analyses.

See "[Electrostatic Field Analysis \(h-Method\)](#)", "[p-Method Electrostatic Analysis](#)", and "[Electric Circuit Analysis](#)" for a description of the other types of electric field analysis.

ANSYS uses Maxwell's equations as the basis for electric field analysis. Refer to "[Electromagnetics](#)" in the [Theory Reference for ANSYS and ANSYS Workbench](#) for details. The primary unknowns (nodal degrees of freedom) that the finite element solution calculates are electric scalar potentials (voltages). Other electric field quantities are then derived from the nodal potentials.

This document describes electric-only field analysis, specifically steady-state current conduction analysis, quasistatic time-harmonic and time-transient electric field analyses, electrostatic field analysis, and electric circuit analysis. Some of the elements described can also be used as coupled-field elements. The [Coupled-Field Analysis Guide](#) discusses coupled-field analyses.

Electric contact is also available in ANSYS. See [Modeling Electric Contact](#) in the [Contact Technology Guide](#) for details.

The following electric field topics are available:

- [Elements Used in Electric Field Analysis](#)

- [Element Compatibility](#)
 - [Current Densities](#)
 - [Steady-State Current Conduction Analysis](#)
 - [Harmonic Quasistatic Electric Analysis](#)
 - [Transient Quasistatic Electric Analysis](#)
 - [Sample Steady-State Conduction Current Analysis](#)
 - [Sample Conductance Calculation](#)
 - [Sample Harmonic Quasistatic Electric Analysis](#)
 - [Sample Transient Quasistatic Electric Analysis](#)
 - [Where to Find Current Conduction Analysis Examples](#)
-
-

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

12.1. Elements Used in Electric Field Analysis

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

The ANSYS program has a large number of elements available for specific types of electric field analysis. The following tables summarize them to make element type selection easier. These elements enable you to perform the following types of analysis:

- Steady-state current conduction analysis
- Quasistatic time-harmonic and time-transient electric field analyses
- Electrostatic field analysis
- Electric circuit analysis

Table 12.1 Conducting Bar Elements

| Element | Dimens. | Shape or Characteristic | DOFs | Usage Notes |
|------------------------|---------|-------------------------|--------------------------------------|---|
| LINK68 | 3-D | Uniaxial, two nodes | Temperature and voltage at each node | Steady-state current conduction analysis; thermal-electric coupled-field analysis |

Table 12.2 2-D PlanarElements

| Element | Dimens. | Shape or Characteristic | DOFs | Usage Notes |
|--------------------------|---------|----------------------------|--------------------------------------|---|
| PLANE67 | 2-D | Quadrilateral, four nodes | Temperature and voltage at each node | Steady-state current conduction analysis; thermal-electric coupled-field analysis |
| PLANE121 | 2-D | Quadrilateral, eight nodes | Voltage at each node | Electrostatic analysis; quasistatic time-harmonic analysis |
| PLANE230 | 2-D | Quadrilateral, eight nodes | Voltage at each node | Steady-state current conduction analysis; quasistatic time-harmonic and time-transient analyses |

Table 12.3 3-D Solid Elements

| Element | Dimens. | Shape or Characteristic | DOFs | Usage Notes |
|---------|---------|-------------------------|------|-------------|
| | | | | |

| | | | | |
|---------------------------------|-----|--------------------------|---|---|
| <u>SOLID5</u> | 3-D | Hexahedral, eight nodes | Up to six at each node; the DOFs are structural displacements, temperature, electric potential, and magnetic scalar potential | Steady-state current conduction analysis; thermal-electric coupled-field analysis or coupled-field electromagnetic analysis |
| <u>SOLID69</u> | 3-D | Hexahedral, eight nodes | Temperature and voltage at each node | Steady-state current conduction analysis; thermal-electric coupled-field analysis |
| <u>SOLID98</u> | 3-D | Tetrahedral, ten nodes | Up to six at each node; the DOFs are structural displacements, temperature, electric potential, and magnetic scalar potential | Steady-state current conduction analysis; thermal-electric coupled-field analysis or coupled-field electromagnetic analysis |
| <u>SOLID122</u> | 3-D | Hexahedral, twenty nodes | Voltage at each node | Electrostatic analysis; quasistatic time-harmonic analysis |
| <u>SOLID123</u> | 3-D | Tetrahedral, ten nodes | Voltage at each node | Electrostatic analysis; quasistatic time-harmonic analysis |
| <u>SOLID127</u> | 3-D | Tetrahedral, ten nodes | Voltage at each node | Electrostatic analysis |
| <u>SOLID128</u> | 3-D | Hexahedral, twenty nodes | Voltage at each node | Electrostatic analysis |
| <u>SOLID231</u> | 3-D | Hexahedral, twenty nodes | Voltage at each node | Steady-state current conduction analysis; quasistatic time-harmonic and time-transient analyses |
| <u>SOLID232</u> | 3-D | Tetrahedral, ten nodes | Voltage at each node | Steady-state current conduction analysis; quasistatic time-harmonic and time-transient analyses |

Table 12.4 Shell Elements

| Element | Dimens. | Shape or Characteristic | DOFs | Usage Notes |
|---------------------------------|---------|---------------------------------|--------------------------------------|---|
| <u>SHELL157</u> | 3-D | Quadrilateral shell, four nodes | Temperature and voltage at each node | Steady-state current conduction analysis; thermal-electric coupled-field analysis |

Table 12.5 Specialty Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|---------------------------------|----------------------|---|--|
| <u>MATRIX50</u> | None (super-element) | Depends on the elements that it includes in its structure | Depends on the included element types |
| <u>INFIN110</u> | 2-D | Four or eight nodes | One per node; this can be a magnetic vector potential, temperature, or electric potential |
| <u>INFIN111</u> | 3-D | Hexahedral, eight or twenty nodes | AX, AY, AZ magnetic vector potential, temperature, electric scalar potential, or magnetic scalar potential |

Table 12.6 General Circuit Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|--------------------------|---------|---|---|
| CIRCU94 | None | Circuit element for use in piezoelectric-circuit analyses, two or three nodes | Voltage at two nodes (plus charge at a third node for an independent voltage source) |
| CIRCU124 | None | General circuit element applicable to circuit simulation, up to six nodes | Up to three at each node; these can be electric potential, current, or electromotive force drop |
| CIRCU125 | None | Diode element used in electric circuit analysis, two nodes | Electric potential |

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

12.2. Element Compatibility

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

Your finite element model may intermix certain elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution (see table below). Electric charge reactions must all be positive or negative.

Table 12.7 Reaction Solutions for Elements with VOLT DOF

| Element | KEYOPT (1) | DOFs | Material Property Input for VOLT DOF | Reaction Solution |
|--------------------------|------------|------------------------|--|---|
| PLANE67 | N/A | TEMP, VOLT | RSVX, RSVY | Electric Current (F label = AMPS) |
| LINK68 | N/A | TEMP, VOLT | RSVX | Electric Current (F label = AMPS) |
| SOLID69 | N/A | TEMP, VOLT | RSVX, RSVY, RSVZ | Electric Current (F label = AMPS) |
| SHELL157 | N/A | TEMP, VOLT | RSVX, RSVY | Electric Current (F label = AMPS) |
| PLANE53 | 1 | VOLT, AZ | RSVX, RSVY | Electric Current (F label = AMPS) |
| SOLID97 | 1 | AX, AY, AX, VOLT | RSVX, RSVY, RSVZ | Electric Current (F label = AMPS) |
| | 4 | AX, AY, AZ, VOLT, Curr | | |
| SOLID117 | 1 | AZ, VOLT | RSVX, RSVY, RSVZ | Electric Current (F label = AMPS) |
| PLANE121 | N/A | VOLT | RSVX, RSVY, PERX, PERY, LSST | Positive or Negative Electric Charge (F label = CHRG) [1] |
| SOLID122 | N/A | VOLT | RSVX, RSVY, RSVZ, PERX, PERY, PERZ, LSST | Positive or Negative Electric Charge (F label = CHRG) [1] |
| SOLID123 | N/A | VOLT | RSVX, RSVY, RSVZ, PERX, PERY, PERZ, LSST | Positive or Negative Electric Charge (F label = CHRG) [1] |
| SOLID127 | N/A | VOLT | PERX, PERY, PERZ | Positive Electric Charge (F label = CHRG) |
| SOLID128 | N/A | VOLT | PERX, PERY, PERZ | Positive Electric Charge (F label = CHRG) |
| PLANE230 | N/A | VOLT | RSVX, RSVY, PERX, PERY, LSST | Electric Current (F label = AMPS) |
| SOLID231 | N/A | VOLT | RSVX, RSVY, RSVZ, PERX, PERY, PERZ, LSST | Electric Current (F label = AMPS) |
| SOLID232 | N/A | VOLT | RSVX, RSVY, RSVZ, PERX, PERY, PERZ, LSST | Electric Current (F label = AMPS) |

| | | | | |
|--------------------------|--------|------------------------------|--------------------------------------|--|
| CIRCU94 | 0-5 | VOLT, CURR | N/A | Negative or Positive Electric Charge (F label = CHRG or AMPS) [2] |
| CIRCU124 | 0-12 | VOLT, CURR, EMF | N/A | Electric Current (F label = AMPS) |
| CIRCU125 | 0 or 1 | VOLT | N/A | Electric Current (F label = AMPS) |
| TRANS109 | 0 or 1 | UX, UY, VOLT | PERX | Positive Electric Charge (F label = CHRG) Mechanical Force (F label = FX, FY) |
| TRANS126 | N/A | UX-VOLT, UY-VOLT, UZ-VOLT | N/A | Electric Current (F label = AMPS) Mechanical Force (F label = FX) |
| PLANE13 | 6 | VOLT, AZ | RSVX, RSVY | Electric Current (F label = AMPS) |
| | 7 | UX, UY, UZ, VOLT | PERX, PERY | Negative Electric Charge (F label = AMPS) |
| SOLID5 | 0 | UX, UY, UZ, TEMP, VOLT, MAG | RSVX, RSVY, RSVZ PERX, PERY, PERZ | Electric Current (F label = AMPS) Negative Electric Charge (F label = AMPS) |
| | 1 | TEMP, VOLT, MAG | RSVX, RSVY, RSVZ | Electric Current (F label = AMPS) |
| | 3 | UX, UY, UZ, VOLT | PERX, PERY, PERZ | Negative Electric Charge (F label = AMPS) |
| | 9 | VOLT | RSVX, RSVY, RSVZ | Electric Current (F label = AMPS) |
| | 0 | UX, UY, UZ, TEMP, VOLT, MAG | RSVX, RSVY, RSVZ PERX, PERY, PERZ | Electric Current (F label = AMPS) Negative Electric Charge (F label = AMPS) |
| | 1 | TEMP, VOLT, MAG | RSVX, RSVY, RSVZ | Electric Current (F label = AMPS) |
| SOLID98 | 3 | UX, UY, UZ, VOLT | PERX, PERY, PERZ | Negative Electric Charge (F label = AMPS) |
| | 9 | VOLT | RSVX, RSVY, RSVZ | Electric Current (F label = AMPS) |
| | N/A | UX, UY, UZ, AX, AY, AZ, VOLT | RSVX, RSVY, RSVZ | Electric Current (F label = AMPS) |
| PLANE223 | 101 | UX, UY, VOLT | RSVX, RSVY | Electric Current (F label = AMPS) |
| | 1001 | UX, UY, VOLT | PERX, PERY, LSST | Positive or Negative Electric Charge (F label = CHRG) [3] |
| | 110 | TEMP, VOLT | RSVX, RSVY, PERX, PERY | Electric Current (F label = AMPS) |
| | 111 | UX, UY, TEMP, VOLT | RSVX, RSVY, PERX, PERY | Electric Current (F label = AMPS) |
| | 1011 | UX, UY, TEMP, VOLT | PERX, PERY, LSST, DPER | Negative Electric Charge (F label = CHRG) |
| SOLID226 | 101 | UX, UY, UZ, VOLT | RSVX, RSVY, RSVZ | Electric Current (F label = AMPS) |
| | 1001 | UX, UY, UZ, VOLT | PERX, PERY, PERZ, LSST | Positive or Negative Electric Charge (F label = CHRG) [2] |
| | 110 | TEMP, VOLT | RSVX, RSVY, RSVZ, PERX, PERY, PERZ | Electric Current (F label = AMPS) |

| | | | | |
|-----------------|------|---------------------------|---------------------------------------|--|
| | 111 | UX, UY, UZ, TEMP, VOLT | RSVX, RSVY, RSVZ, PERX, PERY, PERZ | Electric Current (F label = AMPS) |
| | 1011 | UX, UY, UZ, TEMP, VOLT | PERX, PERY, PERZ, LSST, DPER | Negative Electric Charge (F label = CHRG) |
| <u>SOLID227</u> | 101 | UX, UY, UZ, VOLT | RSVX, RSVY, RSVZ | Electric Current (F label = AMPS) |
| | 1001 | UX, UY, UZ, VOLT | PERX, PERY, PERZ, LSST | Positive or Negative Electric Charge (F label = CHRG) [2] |
| | 110 | TEMP, VOLT | RSVX, RSVY, RSVZ, PERX, PERY, PERZ | Electric Current (F label = AMPS) |
| | 111 | UX, UY, UZ, TEMP, VOLT | RSVX, RSVY, RSVZ, PERX, PERY, PERZ | Electric Current (F label = AMPS) |
| | 1011 | UX, UY, UZ, TEMP, VOLT | PERX, PERY, PERZ, LSST, DPER | Negative Electric Charge (F label = CHRG) |
| <u>INFIN110</u> | 1 | VOLT | PERX, PERY | Positive or Negative Electric Charge (F label = CHRG) [1] |
| <u>INFIN111</u> | 2 | VOLT | PERX, PERY, PERZ | Positive or Negative Electric Charge (F label = CHRG) [1] |

- The following apply to electrostatic elements [PLANE121](#), [SOLID122](#), and [SOLID123](#) and far-field elements [INFIN110](#) and [INFIN111](#):
 - If KEYOPT(6) is set to 0, the reaction solution is positive electric charge.
 - If KEYOPT(6) is set to 1, the reaction solution is negative electric charge.
- The following apply to circuit element [CIRCU94](#), KEYOPT(1) = 0-4:
 - If KEYOPT(6) is set to 0, the analysis type is piezoelectric-circuit and the reaction solution is negative electric charge.
 - If KEYOPT(6) is set to 1, the analysis type is electrostatic-circuit and the reaction solution is positive electric charge.
- The following apply to coupled-field elements [PLANE223](#), [SOLID226](#), and [SOLID227](#), KEYOPT(1) = 1001:
 - If a piezoelectric matrix is specified (TB,PIEZ), the analysis type is piezoelectric and the reaction solution is negative electric charge.
 - If a piezoelectric matrix is not specified, the analysis type is electroelastic and the reaction solution is positive electric charge.

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

12.3. Current Densities

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

The ANSYS output includes various element current densities (JS, JT, and JC item labels). As shown in the following table, their meaning depends upon the type of low-frequency electromagnetic analysis.

Table 12.8 Current Densities in Low-Frequency Analyses

| Current Density Label | Low-Frequency Electric Analysis | Low-Frequency Magnetic Analysis |
|-----------------------|--|---|
| JS | Total element current density. It is the sum of the element conduction and the displacement current densities. It may be used as a source for a subsequent magnetostatic analysis. | Source current density. |
| JT | Element conduction current density. | Total measurable element current density. |
| JC | Nodal conduction current density. | — |

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

12.4. Steady-State Current Conduction Analysis

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

Steady-state current conduction analysis determines the current density and electric potential (voltage) distribution caused by direct current (DC) or potential drop. You can apply two types of loads in this analysis: voltage and electric current. Refer to "[Electromagnetics](#)" in the [Theory Reference for ANSYS and ANSYS Workbench](#) for more information.

A steady-state current conduction analysis is assumed to be linear. That is, the electric current is proportional to the applied voltage.

The procedure for doing a steady-state current conduction analysis consists of three main steps:

1. [Build the model](#).
2. [Apply loads and obtain the solution](#).
3. [Review the results](#).

The next few topics discuss what you must do to perform these steps.

12.4.1. Building the Model

To build the model, you start by specifying the jobname and a title for your analysis, using the following commands or GUI paths:

Command(s): [/FILNAME](#)
[/TITLE](#)

GUI: Utility Menu> File> Change Jobname
 Utility Menu> File> Change Title

If you are using the ANSYS GUI , the next step is to set preferences for an electric analysis: **Main Menu> Preferences> Electromagnetics> Electric**

You *must* set the preference to Electric to ensure that the elements needed for your analysis will be available. (The ANSYS GUI filters element types based on the preference you choose.)

Once you have set the Electric preference, use the ANSYS preprocessor (PREP7) to define the element types, the material properties, and the model geometry. These tasks are common to most analyses. The [Modeling and Meshing Guide](#) explains them in detail.

You can use the following types of elements in a steady-state current conduction analysis:

Table 12.9 Elements Used in a Steady-State Analysis

| Element | Dimens. | Type |
|--------------------------|---------|--|
| LINK68 | 3-D | Two node thermal /electric line |
| PLANE67 | 2-D | Four node thermal/electric quadrilateral |
| PLANE230 | 2-D | Eight node electric quadrilateral |
| SOLID5 | 3-D | Eight node structural/thermal/magnetic/electric hexahedral |
| SOLID69 | 3-D | Eight node thermal//electric hexahedral |
| SOLID98 | 3-D | Ten node structural/thermal/magnetic/electric tetrahedral |
| SOLID231 | 3-D | Twenty node electric hexahedral |
| SOLID232 | 3-D | Ten node electric tetrahedral |
| SHELL157 | 3-D | Four node thermal/electric shell |
| MATRIX50 | 3-D | Superelement |

You must specify electric resistivity values RSVX, RSVY, and RSVZ using the [**MP**](#) command. These properties may be constant or temperature dependent.

12.4.2. Applying Loads and Obtaining a Solution

In this step, you define the analysis type and options, apply loads to the model, specify load step options, and initiate the finite element solution. The next few topics explain how to perform the following tasks:

1. [Enter the SOLUTION Processor.](#)
2. [Define the analysis type.](#)
3. [Define the analysis options.](#)
4. [Apply loads.](#)
5. [Start the solution.](#)
6. [Finish the solution.](#)

12.4.2.1. Entering the SOLUTION Processor

To enter the SOLUTION processor, use either of the following:

Command(s): [**/SOLU**](#)

GUI: Main Menu> Solution

12.4.2.2. Defining Analysis Type

To specify the analysis type, do either of the following:

- In the GUI, choose menu path **Main Menu> Solution> Analysis Type> New Analysis** and choose a

Steady-state analysis.

- If this is a new analysis, issue the command **ANTYPE**,STATIC,NEW.
- If you want to restart a previous analysis (for example, to specify additional loads), issue the command **ANTYPE**,STATIC,REST. You can restart an analysis only if you previously completed a steady-state analysis, and the files Jobname.EMAT, Jobname.ESAV, and Jobname.DB from the previous run are available.

12.4.2.3. Defining Analysis Options

Next, you define which solver you want to use. You can use the sparse solver (default), the frontal solver, the Jacobi Conjugate Gradient (JCG) solver, the Incomplete Cholesky Conjugate Gradient (ICCG) solver, or the Preconditioned Conjugate Gradient solver (PCG).

To select an equation solver, use either of the following:

Command(s): **EOSLV**

GUI: Main Menu> Solution> Analysis Type> Analysis Options

12.4.2.4. Applying Loads

You can apply loads to a steady-state analysis either on the solid model (keypoints, lines, and areas) or on the finite element model (nodes and elements). You can specify several types of loads:

12.4.2.4.1. Current

Electric currents (AMPS) are concentrated nodal loads that you usually specify at model boundaries (the label AMPS is just a load label; it does not indicate the units of measurement). A positive value of current indicates current flowing into the node. For a uniform current density distribution, couple the appropriate nodes in the VOLT degree of freedom, and apply the full current at one of the nodes.

To apply current, use one of the following:

Command(s): **F**

GUI: Main Menu> Solution> Define Loads> Apply> Electric> Excitation> Current

12.4.2.4.2. Voltage (VOLT)

Voltages are DOF constraints that you usually specify at model boundaries to apply a known voltage. A typical approach specifies a zero voltage at one end of the conductor (the "ground" end) and a desired voltage at the other end.

To apply voltage, use the following command or GUI path:

Command(s): **D**

GUI: Main Menu> Solution> Define Loads> Apply> Electric> Boundary> Voltage

You can also apply current and voltage loads using the independent current and voltage source options of [CIRCU124](#). For more information, refer to "[Electric Circuit Analysis](#)".

Optionally, you can use other commands to apply loads to a steady-state analysis, and you also can specify output controls as load step options. For information about using these commands to apply loads and about the load step options available for steady-state analysis, see "[Alternative Analysis Options and Solution Methods](#)".

12.4.2.5. Starting the Solution

In this step, you initiate the solution for all load steps using one of the following:

Command(s): [**SOLVE**](#)

GUI: Main Menu> Solution> Solve> Current LS

12.4.2.6. Finishing the Solution

To leave the SOLUTION processor, use either of the following:

Command(s): [**FINISH**](#)

GUI: Main Menu> Finish

12.4.3. Reviewing Results

The program writes results from a steady-state current conduction analysis to the results file, Jobname.RTH (or to Jobname.RST if other degrees of freedom are available besides VOLT). Results include the data listed below:

Primary data: Nodal voltages (VOLT).

Derived data:

- Nodal electric field (EFX, EFY, EFZ, EFSUM).
- Nodal conduction current densities (JCX, JCY, JCZ, JCSUM). Supported only by [PLANE230](#), [SOLID231](#), and [SOLID232](#).
- Element conduction current densities (JSX, JSY, JSZ, JSSUM, JTX, JTY, JTZ, JTSUM).
- Element Joule heat (JHEAT).
- Nodal reaction currents.

You can review analysis results in POST1, the general postprocessor. To access the postprocessor, choose one of the following:

Command(s): [**/POST1**](#)

GUI: Main Menu> General Postproc

For a complete description of all postprocessing functions, see the [Basic Analysis Guide](#).

12.4.3.1. Reviewing Results in POST1

To review results in POST1, the ANSYS database must contain the same model for which the solution was calculated. Also, the results file (`Jobname.RTH` or `Jobname.RST`) must be available.

To read results at the desired time point into the database, use either of the following:

Command(s): SET,,,TIME

GUI: Utility Menu> List> Results> Load Step Summary

If you specify a time value for which no results are available, the program performs linear interpolation to calculate the results at that time.

To identify the results data you want, use a combination of a label and a sequence number or component name.

You can now review the results by obtaining graphics displays and tabular listings. To obtain these, use the following:

Table 12.10 Reviewing Results

| Step | Commands | GUI Path |
|----------------------------------|----------------------------------|--|
| Produce contour displays. | <u>PLESOL</u> , <u>PLNSOL</u> | Main Menu> General Postproc> Plot Results> Contour Plot> Element Solution Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu |
| Produce vector (arrow) displays. | <u>PLVECT</u> | Main Menu> General Postproc> Plot Results> Vector Plot> Predefined Main M |

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

12.5. Harmonic Quasistatic Electric Analysis

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

A harmonic electric analysis determines the effects of alternating current (AC), charge or voltage excitation in electric devices. In this analysis, the time-harmonic electric and magnetic fields are uncoupled, and the electromagnetic field can be treated as quasistatic. Eddy currents are considered to be negligible, and the electric field is derived from the electric scalar potential. Capacitive effects and displacement current are taken into account. Refer to "Electromagnetics" in the [Theory Reference for ANSYS and ANSYS Workbench](#) for more information.

You can use this analysis to determine the voltage, electric field, electric flux density, and electric current density distributions in an electric device as a function of frequency in response to time-harmonic loading.

The procedure for doing a harmonic quasistatic analysis consists of three main steps:

1. [Build the model.](#)
2. [Apply loads and obtain the solution.](#)
3. [Review the results.](#)

The next few topics discuss what you must do to perform these steps.

12.5.1. Building the Model

To build the model, you first specify a jobname and a title for your analysis as described in [Steady-State Current Conduction Analysis](#). If you are using the ANSYS GUI, you set preferences for an electric analysis. You then use the ANSYS preprocessor (PREP7) to define the element types, the material properties, and the model geometry.

To perform a current-based harmonic quasistatic analysis, you can use the following types of elements:

Table 12.12 Elements Used in a Current-Based Harmonic Analysis [1]

| Element | Dimens. | Type |
|--------------------------|---------|-----------------------------------|
| PLANE230 | 2-D | Eight node electric quadrilateral |
| SOLID231 | 3-D | Twenty node electric hexahedral |
| SOLID232 | 3-D | Ten node electric tetrahedral |

1. The reaction solution is current.

To perform a charge-based harmonic quasistatic analysis, you can use the following types of elements:

Table 12.13 Elements Used in a Charge-Based Harmonic Analysis [1]

| Element | Dimens. | Type |
|--------------------------|---------|--|
| PLANE121 | 3-D | Eight node electrostatic quadrilateral |
| SOLID122 | 3-D | Twenty node electrostatic hexahedral |
| SOLID123 | 3-D | Ten node electrostatic tetrahedral |

1. The reaction solution is charge.

The default system of units is MKS. In the MKS system of units, free-space permittivity is set to 8.85e-12 Farads/meter. To specify your own system of units and free-space permittivity use one of the following:

Command(s): [EMUNIT](#)

GUI: Main Menu>Preprocessor>Material props>Electromag Units

To model resistive and capacitive effects, a harmonic electric analysis requires the specification of electrical resistivity and electric permittivity, respectively. Define electrical resistivity values as RSVX, RSVY, and RSVZ on the MP command. Define relative electric permittivity values as PERX, PERY, and PERZ on the [MP](#) command.

You can specify losses by defining electrical resistivity (RSVX, RSVY, RSVZ) or a loss tangent (LSST) on the [MP](#) command. Resistivity and loss tangent effects are additive.

These properties may be constant or temperature dependent.

12.5.2. Applying Loads and Obtaining a Solution

In this step, you define the analysis type and options, apply loads to the model, specify load step options, and initiate the finite element solution. The next few topics explain how to perform the following tasks:

1. [Enter the SOLUTION processor.](#)
2. [Define the analysis type.](#)
3. [Define the analysis options.](#)
4. [Apply loads.](#)
5. [Start the solution.](#)
6. [Finish the solution.](#)

12.5.2.1. Entering the SOLUTION Processor

To enter the SOLUTION processor, use either of the following:

Command(s): [/SOLU](#)

GUI: Main Menu> Solution

12.5.2.2. Defining the Analysis Type

To specify the analysis type, do either of the following:

- In the GUI, choose menu path **Main Menu> Solution> Analysis Type> New Analysis** and choose a Harmonic analysis.
- If this is a new analysis, issue the command **ANTYPE,HARMONIC,NEW**.
- If you want to restart a previous analysis (for example, to specify additional loads), issue the command **ANTYPE,HARMONIC,REST**. You can restart an analysis only if you previously completed a harmonic analysis, and the files Jobname.EMAT, Jobname.ESAV, and Jobname.DB from the previous run are available.

12.5.2.3. Defining Analysis Options

Next, you define which solution method and which solver you want to use. Harmonic electric analyses require the full solution method. To select a solution method, use one of the following:

Command(s): **HRNOPT,FULL**

GUI: Main Menu> Solution> Analysis Type> New Analysis> Harmonic

You can use the sparse solver (default), the frontal solver, the Jacobi Conjugate Gradient (JCG) solver, the Incomplete Cholesky Conjugate Gradient (ICCG) solver, or the Preconditioned Conjugate Gradient solver (PCG). To select an equation solver, use one of the following:

Command(s): **EOSLV**

GUI: Main Menu> Solution> Analysis Type> Analysis Options

To specify the frequency range, use any of the following:

Command(s): **HARFRO**

GUI: Main Menu> Preprocessor> Loads> Load Step opts> Time/Frequency
Main Menu> Solution> Loads> Load Step opts> Time/Frequency

To specify the number of harmonic solutions within the load step, use either of the following:

Command(s): **NSUBS**

GUI: Main Menu> Preprocessor> Loads> Load Step Opt> Time/Frequenc> Freq & and Substps
Main Menu> Solution> Load Step Opt> Time/Frequenc> Freq & Substps

When specifying multiple substeps within a load step, you need to indicate whether the loads are to be ramped or stepped. The **KBC** command is used for this purpose: **KBC,0** indicates ramped loads (default), and **KBC,1** indicates stepped loads.

Command(s): **KBC**

GUI: Main Menu> Solution> Load Step Opt> Time/Frequenc> Time and Substps (or Time & Time Step)

Main Menu> Solution> Load Step Opt> Time/Frequency> Time and Substeps (or Time & Time Step)

To specify results data for the printed output file (`Jobname.OUT`), use one of the following:

Command(s): OUTPR

GUI: **Main Menu> Preprocessor> Loads> Load Step Opt> Output Ctrls> Solu Printout**
Main Menu> Solution> Load Step Opt> Output Ctrls> Solu Printout

You can also control the solution items sent to the results file (`Jobname.RTH`). By default, the ANSYS program writes only the last substep of each load step to the results file. If you want all substeps (that is, the solution at all time substeps) on the results file, use one of the following to specify a frequency or ALL or 1.

Command(s): OUTRES

GUI: **Main Menu> Preprocessor> Loads> Load Step Opt> Output Ctrls> DB/Results File**
Main Menu> Solution> Loads> Load Step Opt> Output Ctrls> DB/Results File

12.5.2.4. Applying Loads

You can apply loads in a harmonic analysis either on the solid model (keypoints, lines, and areas) or on the finite element model (nodes and elements). The type of loads you can specify depends on the element type chosen for a harmonic analysis.

Table 12.14 Load Types

| Analysis | Element Types | Loads |
|------------------------|--|--|
| Current-Based Analysis | PLANE230 , SOLID231 , SOLID232 | Current Voltage |
| Charge-Based Analysis | PLANE121 , SOLID122 , SOLID123 | Charge Surface charge density Volume charge density Voltage |

12.5.2.4.1. Current

Electric currents (AMPS) are concentrated nodal loads that you usually specify at model boundaries (the label AMPS is just a load label; it does not indicate the units of measurement). A positive value of current indicates current flowing into the node. For a uniform current density distribution, couple the appropriate nodes in the VOLT degree of freedom, and apply the full current at one of the nodes.

To apply current, use one of the following:

Command(s): F

GUI: **Main Menu> Solution> Define Loads> Apply> Electric> Excitation> Current**

You can also apply current loads using the independent current source option of [CIRCU124](#). For more information, refer to ["Electric Circuit Analysis"](#).

12.5.2.4.2. Charge

Electric charges (CHRG) are concentrated nodal force loads. To apply them, use the following command or GUI path:

Command(s): [**F**](#)

GUI: **Main Menu> Solution> Define Loads> Apply> Electric> Excitation> Charge> On Nodes**

12.5.2.4.3. Voltage (VOLT)

Voltages are DOF constraints that you usually specify at model boundaries to apply a known voltage. A typical approach specifies a zero voltage at one end of the conductor (the "ground" end) and a desired voltage at the other end.

To apply voltage, use the following command or GUI path:

Command(s): [**D**](#)

GUI: **Main Menu> Solution> Define Loads> Apply> Electric> Boundary> Voltage**

You can also apply voltage loads using the independent voltage source option of [CIRCU124](#). For more information, refer to ["Electric Circuit Analysis"](#).

12.5.2.5. Starting the Solution

In this step, you initiate the solution for all load steps using one of the following:

Command(s): [**SOLVE**](#)

GUI: **Main Menu> Solution> Solve> Current LS**

12.5.2.6. Finishing the Solution

To leave the SOLUTION processor, use either of the following:

Command(s): [**FINISH**](#)

GUI: **Main Menu> Finish**

12.5.3. Reviewing Results

The program writes results from a harmonic electric analysis to the results file, `Jobname.RTH`. Results include the data listed below:

Primary data: Nodal DOF (VOLT).

Derived data:

Note

Some output quantities depend on the element type used in the analysis.

- Nodal electric field (EFX, EFY, EFZ, EFSUM).
- For a current-based analysis using electric elements, nodal conduction current densities (JCX, JCY, JCZ, JCSUM).
- For a charge-based analysis using electrostatic elements, nodal electric flux densities (DX, DY, DZ, DSUM).
- Element current densities (JSX, JSY, JSZ, JSSUM). This output item represents the total (that is, sum of conduction and displacement current densities). It can be used as a source for a subsequent magnetic analysis.
- Element conduction current densities (or total measurable current density) (JTX, JTY, JTZ, JTSUM).
- Element Joule heat generation rate per unit volume (JHEAT). This is a time-averaged value.
- Element stored electric energy (SENE). This is a time-averaged value.
- For a current-based analysis using electric elements, nodal reaction currents.
- For a charge-based analysis using electrostatic elements, nodal reaction charges.

You can review analysis results in POST1, the general postprocessor, or in POST26, the time-history postprocessor. To access the general postprocessor, choose one of the following:

Command(s): [/POST1](#)

GUI: Main Menu> General Postproc

To access the time-history postprocessor, choose one of the following:

Command(s): [/POST26](#)

GUI: Main Menu> TimeHist Postproc

The following table summarizes the applicable labels for the /POST1 and /POST26 commands.

Table 12.15 Command Labels

| Output Quantity | Label | Command(s) | Analysis | |
|----------------------------------|-------|---|-------------------|------------------|
| | | | Current-based [1] | Charge-based [2] |
| Nodal DOF | VOLT | <u>PRNSOL</u> , <u>PLNSOL</u> , <u>ETABLE</u> , <u>NSOL</u> , | Y | Y |
| Nodal electric field | EF | <u>PRNSOL</u> , <u>PLNSOL</u> , | Y | Y |
| Nodal conduction current density | JC | | Y | - |

| | | | | |
|--|--------|---|---|---|
| Nodal electric flux density | D | PRESOL , PLESOL , PRVECT , PLVECT , ETABLE , ESOL | - | Y |
| Element total current density | JS [3] | PRESOL , PLESOL , PRVECT , PLVECT , ETABLE , ESOL | Y | Y |
| Element conduction current density | JT [3] | PRESOL , PLESOL , PRVECT , PLVECT , ETABLE , ESOL | Y | Y |
| Element Joule heat generation rate per unit volume (time-averaged) | JHEAT | PRESOL , PLESOL , ETABLE , ESOL | Y | Y |
| Element stored electric energy (time-averaged) | SENE | PRESOL , PLESOL , ETABLE , ESOL | Y | Y |
| Nodal reaction current | AMPS | RFORCE , PRRFOR , PRRSOL , PRESOL , PLESOL | Y | - |
| Nodal reaction charge | CHRG | RFORCE , PRRFOR , PRRSOL , PRESOL , PLESOL | - | Y |

1. [PLANE230](#), [SOLID231](#), and [SOLID232](#) are current-based.
2. [PLANE121](#), [SOLID122](#), and [SOLID123](#) are charge-based.
3. Refer to [Table 12.8: "Current Densities in Low-Frequency Analyses"](#) for the meaning of this label.

For a complete description of all postprocessing functions, see "[The General Postprocessor \(POST1\)](#)" and "[The Time-History Postprocessor \(POST26\)](#)" in the [Basic Analysis Guide](#).

12.5.3.1. Reviewing Results in POST1

To review results in POST1, the ANSYS database must contain the same model for which the solution was calculated. Also, the results file (`Jobname.RTH`) must be available.

The procedures for reviewing POST1 harmonic electric analysis results are identical to the procedures described in [Steady-State Current Conduction Analysis](#) with the following exception. Results from a harmonic electric analysis are complex and consist of real and imaginary components. Set KIMG=0 or KIMG=1 on the SET command to read the real or imaginary results respectively.

12.5.3.2. Reviewing Results in POST26

To review results in POST26, the time-history postprocessor, the ANSYS database must contain the same model for which the solution was calculated, and the results file (`Jobname.RTH`) must be available. If the model is not in the database, restore it using one of the following:

Command(s): [RESUME](#)

GUI: Utility Menu> File> Resume Jobname.db

Then use one of the following to read in the desired set of results.

Command(s): SET**GUI:** Utility Menu> List> Results> Load Step Summary

POST26 works with tables of result item versus frequency, known as variables. Each variable is assigned a reference number, with variable number 1 reserved for frequency. Therefore the first things you need to do is define the variables using the following commands or GUI paths.

Table 12.16 Defining Variables

| Step | Command | GUI Path |
|---------------------------------|-------------------------------|--|
| Define primary data variables | <u>NSOL</u> | Main Menu> TimeHist Postpro> Define Variables |
| Define derived data variables | <u>ESOL</u> | Main Menu> TimeHist Postpro> Define Variables |
| Define reaction data variables. | <u>RFORCE</u> | Main Menu> TimeHist Postpro> Define Variables |

Once you have defined these variables, you can graph or list them (versus time or any variable) using the following commands or GUI paths.

Table 12.17 Graphing and Listing Variables

| Step | Command | GUI Path |
|-------------------------------------|-------------------------------|---|
| To graph variables. | <u>PLVAR</u> | Main Menu> TimeHist Postpro> Graph Variables |
| To list variables. | <u>PRVAR</u> | Main Menu> TimeHist Postpro> List Variables |
| To list only the extreme variables. | <u>EXTREM</u> | Main Menu> TimeHist Postpro> List Extremes |

POST26 offers many other functions, such as performing math operations among variables, moving variables into array parameters, etc. For more information, see "[The Time-History Postprocessor \(POST26\)](#)" in the [Basic Analysis Guide](#).

By reviewing the time-history results at strategic points throughout the model, you can identify the critical time points for further POST1 postprocessing.

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

12.6. Transient Quasistatic Electric Analysis

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

A transient electric analysis determines the effects of time-dependent current or voltage excitation in electric devices. In this analysis, the time-varying electric and magnetic fields are uncoupled, and the electromagnetic field can be treated as quasistatic. Eddy currents are considered to be negligible, and the electric field is derived from the electric scalar potential. A transient electric analysis is assumed to be linear. Refer to "[Electromagnetics](#)" in the [Theory Reference for ANSYS and ANSYS Workbench](#) for more information.

You can use this analysis to determine the voltage, electric field, and electric current density distributions in an electric device as a function of time in response to time-dependent loading. The time scale of the loading is such that the capacitive effects and displacement current are considered to be important. If they are not important, you might be able to use a steady-state current conduction analysis instead.

The procedure for doing a transient quasistatic analysis consists of three main steps:

1. [Build the model.](#)
2. [Apply loads and obtain the solution.](#)
3. [Review the results.](#)

The next few topics discuss what you must do to perform these steps.

12.6.1. Building the Model

To build the model, you first specify a jobname and a title for your analysis as described in [Steady-State Current Conduction Analysis](#). If you are using the ANSYS GUI, you set preferences for an electric analysis. You then use the ANSYS preprocessor (PREP7) to define the element types, the material properties, and the model geometry.

You can use the following types of elements in a transient electric analysis:

Table 12.18 Elements Used in a Transient Analysis

| Element | Dimens. | Type |
|--------------------------|---------|-----------------------------------|
| PLANE230 | 2-D | Eight node electric quadrilateral |
| SOLID231 | 3-D | Twenty node electric hexahedral |
| SOLID232 | 3-D | Ten node electric tetrahedral |

The default system of units is MKS. In the MKS system of units, free-space permittivity is set to 8.85e-12 Farads/meter. To specify your own system of units and free-space permittivity use one of the following:

Command(s): [**EMUNIT**](#)

GUI: Main Menu>Preprocessor>Material props>Electromag Units

To model resistive and capacitive effects, a transient electric analysis requires the specification of electrical resistivity and electric permittivity, respectively. Define electrical resistivity values as RSVX, RSVY, and RSVZ on the [**MP**](#) command. Define relative electric permittivity values as PERX, PERY, and PERZ on the [**MP**](#) command. These properties may be constant or temperature dependent.

12.6.2. Applying Loads and Obtaining a Solution

In this step, you define the analysis type and options, apply loads to the model, specify load step options, and initiate the finite element solution. The next few topics explain how to perform the following tasks:

1. [Enter the SOLUTION processor.](#)
2. [Define the analysis type.](#)
3. [Define the analysis options.](#)
4. [Apply loads.](#)
5. [Start the solution.](#)
6. [Finish the solution.](#)

12.6.2.1. Entering the SOLUTION Processor

To enter the SOLUTION processor, use either of the following:

Command(s): [**/SOLU**](#)

GUI: Main Menu> Solution

12.6.2.2. Defining the Analysis Type

To specify the analysis type, do either of the following:

- In the GUI, choose menu path **Main Menu> Solution> Analysis Type> New Analysis** and choose a Transient analysis.
- If this is a new analysis, issue the command [**ANTYPE,TRANSIENT,NEW**](#).
- If you want to restart a previous analysis (for example, to specify additional loads), issue the command [**ANTYPE,TRANSIENT,REST**](#). You can restart an analysis only if you previously completed a transient analysis, and the files Jobname.EMAT, Jobname.ESAV, and Jobname.DB from the previous run are available.

12.6.2.3. Defining Analysis Options

Next, you define which solution method and which solver you want to use. Transient electric analyses require the full solution method. To select a solution method, use one of the following:

Command(s): [TRNOPT](#),[FULL](#)

GUI: Main Menu> Solution> Analysis Type> New Analysis>Transient

You can use the sparse solver (default), the frontal solver, the Jacobi Conjugate Gradient (JCG) solver, the Incomplete Cholesky Conjugate Gradient (ICCG) solver, or the Preconditioned Conjugate Gradient solver (PCG). To select an equation solver, use one of the following:

Command(s): [EOSLV](#)

GUI: Main Menu> Solution> Analysis Type> Analysis Options

To specify the time at the end of a load step, use any of the following:

Command(s): [TIME](#)

GUI: Main Menu>Preprocessor>Loads>Load Step opts>Time/Frequency
Main Menu>Solution>Loads>Load Step opts>Time/Frequency

The integration time step is the time increment used in the time integration scheme. It determines the accuracy of your solution. The smaller the time step size, the higher the accuracy. The size of the first integration time step following any large step change in loading conditions is especially critical. You can reduce inaccuracies by reducing the integration time step size. You can specify it directly via the [DELTIM](#) command or indirectly via the [NSUBST](#) command.

Command(s): [DELTIM](#)

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time & Time Step
Main Menu> Solution> Load Step Opts> Time/Frequenc> Time & Time Step

Command(s): [NSUBS](#)

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time and Substps
Main Menu> Solution> Load Step Opts> Time/Frequenc> Time and Substps

When specifying multiple substeps within a load step, you need to indicate whether the loads are to be ramped or stepped. The [KBC](#) command is used for this purpose: [KBC](#),0 indicates ramped loads (default), and [KBC](#),1 indicates stepped loads.

Command(s): [KBC](#)

GUI: Main Menu> Solution> Load Step Opts> Time/Frequenc> Time and Substps (or Time & Time Step)
Main Menu> Solution> Load Step Opts> Time/Frequenc> Time and Substps (or Time & Time Step)

To specify results data for the printed output file (`Jobname.OUT`), use one of the following:

Command(s): [OUTPR](#)

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Output Ctrls> Solu Printout
Main Menu> Solution> Load Step Opts> Output Ctrls> Solu Printout

You can also control the solution items sent to the results file (`Jobname.RTH`). By default, the ANSYS program writes only the last substep of each load step to the results file. If you want all substeps (that is, the solution at all time substeps) on the results file, use one of the following to specify a frequency or ALL or 1.

Command(s): OUTRES

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Output Ctrl> DB/Results File
GUI: Main Menu> Solution> Loads> Load Step Opts> Output Ctrl> DB/Results File

12.6.2.4. Applying Loads

You can apply loads in a transient analysis either on the solid model (keypoints, lines, and areas) or on the finite element model (nodes and elements). You can specify current and voltage loads. The procedures and GUI paths you use to apply these loads are identical to those described in [Steady-State Current Conduction Analysis](#).

You can also apply current and voltage loads using the independent current and voltage source options of [CIRCU124](#). For more information, refer to ["Electric Circuit Analysis"](#).

12.6.2.5. Starting the Solution

In this step, you initiate the solution for all load steps using one of the following:

Command(s): SOLVE

GUI: Main Menu> Solution> Solve> Current LS

12.6.2.6. Finishing the Solution

To leave the SOLUTION processor, use either of the following:

Command(s): FINISH

GUI: Main Menu> Finish

12.6.3. Reviewing Results

The program writes results from a transient electric analysis to the results file, `Jobname.RTH`. Results include the data listed below:

Primary data: Nodal DOF (VOLT).

Derived data:

- Nodal electric field (EFX, EFY, EFZ, EFSUM).
- Nodal conduction current densities (JCX, JCY, JCZ, JCSUM).
- Element current densities (JSX, JSY, JSZ, JSSUM). This output item represents the total (that is, the sum of conduction and displacement current densities). It can be used as a source for a subsequent magnetic analysis.
- Element conduction current densities (or total measurable current density) (JTX, JTY, JTZ, JTSUM).

- Element Joule heat generation rate per unit volume (JHEAT).
- Element stored electric energy (SENE).
- Nodal reaction currents.

You can review analysis results in POST1, the general postprocessor, or in POST26, the time-history postprocessor. To access the general postprocessor, choose one of the following:

Command(s): [/POST1](#)

GUI: Main Menu> General Postproc

To access the time-history postprocessor, choose one of the following:

Command(s): [/POST26](#)

GUI: Main Menu> TimeHist Postproc

For a complete description of all postprocessing functions, see "[The General Postprocessor \(POST1\)](#)" and "[The Time-History Postprocessor \(POST26\)](#)" in the [Basic Analysis Guide](#).

12.6.3.1. Reviewing Results in POST1

To review results in POST1, the ANSYS database must contain the same model for which the solution was calculated. Also, the results file (`Jobname.RTH`) must be available.

The procedures for reviewing POST1 transient electric analysis results are identical to the procedures described in [Steady-State Current Conduction Analysis](#).

12.6.3.2. Reviewing Results in POST26

To review results in POST26, the time-history postprocessor, the ANSYS database must contain the same model for which the solution was calculated, and the `Jobname.RTH` file (the results file) must be available. If the model is not in the database, restore it using one of the following:

Command(s): [RESUME](#)

GUI: Utility Menu> File> Resume Jobname.db

The procedures for reviewing POST26 transient electric analysis results are identical to the procedures described in [Harmonic Quasistatic Electric Analysis](#). Variable number 1 is reserved for time instead of frequency.

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

12.7. Sample Steady-State Conduction Current Analysis

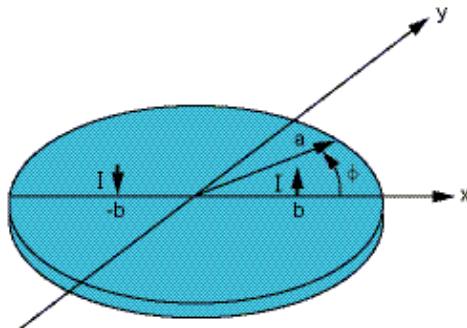
www.kxcad.net Home > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

The following is an example of how to perform a steady-state conduction current analysis by issuing ANSYS commands. You can also perform the analysis through the ANSYS GUI menus.

12.7.1. Problem Description

A current I is applied to a thin disk of radius $r = a$. As shown in the following figure, the current enters and leaves via point electrodes located at $r = b$, $\phi = 0$ and $r = b$, $\phi = \pi$. Find the potential and dc-current distributions in the disk.

Figure 12.3 Conducting Disk with Current Loading



The geometric and electrical parameters are:

Radius $a = 20$ cm

Distance from the center to the point electrodes $b = 10$ cm

Applied Current $I = 1$ mA

Disk Resistivity $\rho = 100 \Omega\text{m}$

To obtain an accurate field distribution, the disc area is densely meshed with triangle-shaped [PLANE230](#) electric elements. A VOLT degree of freedom constraint is applied to the center of the disk. Current loads are applied as concentrated nodal loads. A cylindrical coordinate system is used.

12.7.2. Results

Table 12.19 Electric Potential at Points with Coordinates $r = a$ and ϕ

| ϕ (degrees) | Electric Potential (V) | |
|------------------|------------------------|------------|
| | Computed | Target [1] |
| 0 | -0.03501 | -0.03497 |
| 10 | -0.03399 | -0.03392 |
| 20 | -0.03124 | -0.03110 |
| 30 | -0.02708 | -0.02716 |
| 40 | -0.02269 | -0.02271 |

| | | |
|----|----------|----------|
| 50 | -0.01814 | -0.01810 |
| 60 | -0.01360 | -0.01349 |
| 70 | -0.00880 | -0.00894 |
| 80 | -0.00438 | -0.00445 |
| 90 | 0.0 | 0.0 |

1. Problem # 286, p.137 in N.N. Lebedev, I.P. Skalskaya, Y.S. Ufland, "Worked problems in Applied Mathematics," Dover Publications, Inc., NY (1979).

The following figures display the potential and dc-current distributions in the disk.

Figure 12.4 Potential Distribution

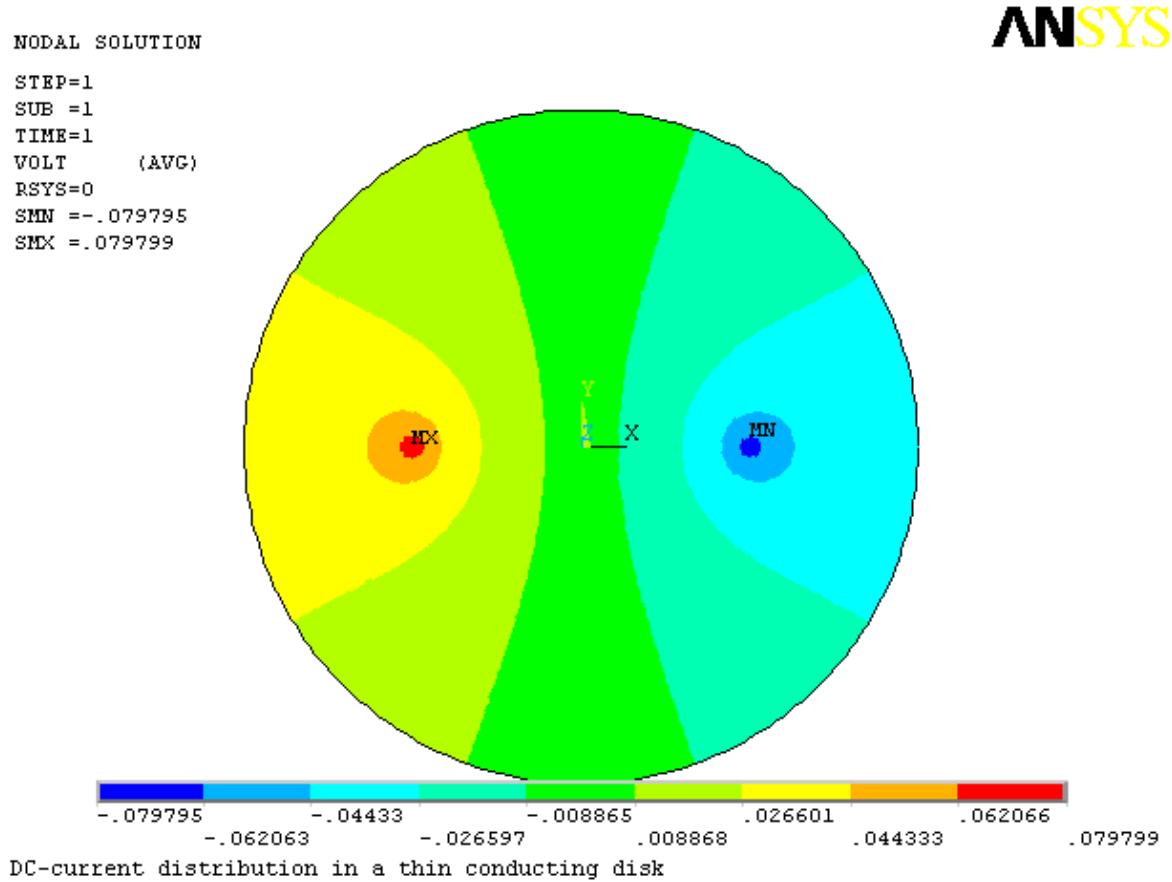
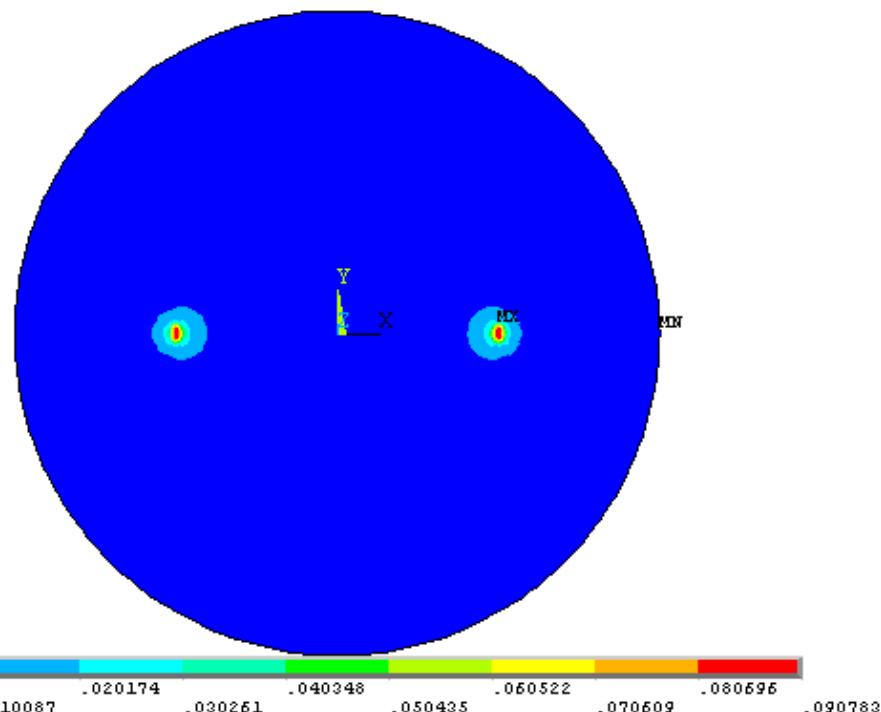


Figure 12.5 Current Distribution

```
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
JCSUM (AVG)
RSYS=0
SMN =.299E-06
SMX =.090783
```



DC-current distribution in a thin conducting disk

12.7.3. Command Listing

You can perform this example steady-state analysis using the ANSYS commands shown below. Text prefaced by an exclamation point (!) is a comment.

```
/batch,list
/title, DC-current distribution in a thin conducting disk
a=20.e-2           ! disk radius, m
b=10.e-2           ! electrode distance from the center, m
I=1.e-3            ! current, A
rho=100             ! resistivity, Ohm*m
/nopr

/PREP7
! Model and meshing
et,1,PLANE230      ! electric element type
mp,rsvx,1,rho       ! resistivity

cyl4,0,0,0,0,a,360   ! circular area
esize,,64
msha,1,2-D           ! mesh with triangles
amesh,1

! Boundary conditions
csys,1               ! cylindrical coordinate system
d,node(0,0,0),volt,0 ! ground center node

! Nodal current loads
f,node(b,0,0),amps,-I
f,node(b,180,0),amps,I
```

```
fini

/solu
antype,static           ! steady-state current conduction
solve
fini

/post1
plnsol,volt             ! plot electric potential
plnsol,jc,sum            ! plot current density vector magnitude

*dim,value,,10,2
*dim,coord,,10,2

*do,i,1,10
! coordinates of solution point
r=a $ phi=(i-1)*10
coord(i,1)=r
coord(i,2)=phi
! ANSYS solution
value(i,1)=volt(node(a,(i-1)*10,0))
*enddo
! Analytical solution
*vfill,value(1,2),data,-0.034969,-0.033923,-0.031098,-0.027163,-0.022709
*vfill,value(6,2),data,-0.018095,-0.013485,-0.008937,-0.004451,0.

/com,----- VOLT SOLUTION -----
/com,
/com,      R      |      PHI     |      ANSYS      |      TARGET
/com,
*vwrite,coord(1,1),coord(1,2),value(1,1),value(1,2),
(1X,' ',F6.1,' ',F6.1,' ',F11.6,' ',F11.6)
/com,-----
fini
```

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

12.8. Sample Conductance Calculation

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

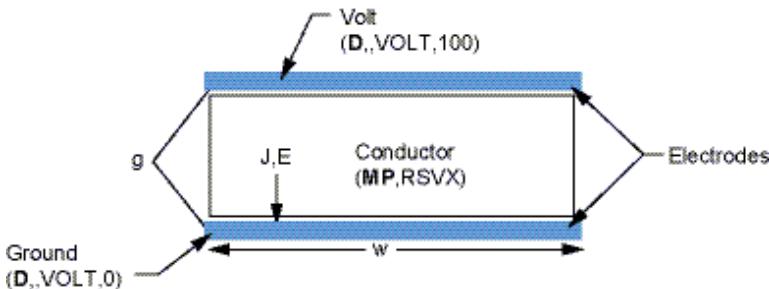
This problem evaluates conductance between parallel plate electrodes. The following is an example of how to conduct the analysis by issuing ANSYS commands. You can also perform the analysis through the ANSYS GUI menus.

12.8.1. Problem Description

In this example, the objective is to compute the self and mutual conductance coefficients between two parallel plate electrodes.

In this example, the model consists of two parallel plate electrodes, 3 m in length, with a gap of 2 m and a conductivity of 10. One electrode represents the ground, and 100 volts is applied to the other electrode. The target conductance is 15.

Figure 12.6 Problem Geometry



12.8.2. Command Listing

```
/title, Conductance between parallel plate electrodes using GMATRIX

/com Input data
/com sigma : conductivity
/com g      : gap between electrodes
/com w      : electrode width
/com l      : length
/com
/com Target
/com
/com G = sigma w l / g : conductance between parallel plate electrodes
!
g=2                      ! gap between electrodes
w=3                      ! electrode width
l=1                      ! length
sigma=10                  ! conductivity
!
gt=sigma*w*l/g          ! target conductance
!
/prep7
```

```
et,1,230
mp,rsvx,1,1/sigma
!
n,1,0,0
n,2,w,0
n,3,w,g
n,4,0,g
e,1,2,3,4
!
nsel,s,loc,y,g           ! electrode
cm,compl,node
d,all,volt,100
!
nsel,s,loc,y,0           ! ground
cm,comp2,node
d,all,volt,0
!
nsel,all
!
gmatrix,1,'comp',2,0      ! compute conductance matrix
gc=gmatrix(1,1,1)          ! computed conductance
finish
```

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

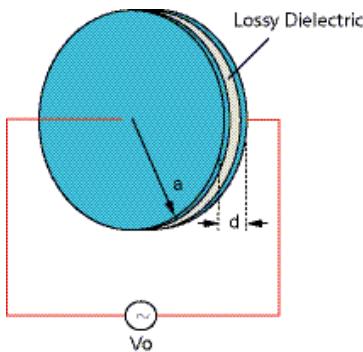
12.9. Sample Harmonic Quasistatic Electric Analysis

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

12.9.1. Problem Description

A time-harmonic voltage of amplitude V_0 is applied to a barium titanate parallel plate capacitor with circular plates of radius a and dielectric thickness d . The dielectric has relative permittivity ϵ_r and loss tangent $\tan\delta$. Perform a harmonic analysis to determine the capacitor admittance (Y) in the frequency (f) range of 0 to 1 MHz, and find the dissipated power at 1 MHz.

Figure 12.7 Parallel Plate Capacitor with Time-Harmonic Voltage Load



The geometric and electrical parameters are:

Radius $a = 9$ cm

Thickness $d = 0.1$ cm

Relative Permittivity $\epsilon_r = 1143$

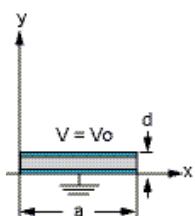
Loss tangent $\tan\delta = 0.0105$

Voltage amplitude $V_0 = 50$ volts

Frequency $f = 0$ to 1 MHz

The capacitor is modeled using the axisymmetric option of [PLANE230](#) electric elements. Electrodes are defined by coupling VOLT degrees of freedom on the major surfaces of the capacitor. The bottom electrode is grounded, and a voltage load V_0 is applied to the top electrode.

Figure 12.8 Axisymmetric Model



Electric admittance (Y) is calculated at ten frequencies between 0 and 1 MHz using the reaction current on the loaded electrode. Results are compared with target values obtained from the following analytical expression:

$$Y = 2\pi f C (\tan\delta + j)$$

where:

$$C = \frac{\epsilon_r \epsilon_0 \pi a^2}{d}$$

and $j = \text{imaginary unit}$.

Power dissipation at $f = 1$ MHz is calculated in /POST1 by summing up the Joule heat rates over the elements. The result is compared with the following analytical expression for the time-average power dissipated in the capacitor:

$$P_d = \pi f V_0^2 C \tan \delta$$

12.9.2. Results

As shown in the following table, the ANSYS electric admittance (Y) results agree with those from the analytical expression given above.

Table 12.20 Electric Admittance (Y)

| Frequency (MHz) | Admittance Amplitude (S) [1] |
|-----------------|------------------------------|
| 0.0 | 0.1618 |
| 0.2 | 0.3236 |
| 0.3 | 0.4855 |
| 0.4 | 0.6473 |
| 0.5 | 0.8091 |
| 0.6 | 0.9709 |
| 0.7 | 1.1327 |
| 0.8 | 1.2945 |
| 0.9 | 1.4564 |
| 1.0 | 1.6182 |

1. The phase angle is 89.4 degrees at all ten frequencies.

The power dissipated at a frequency of 1 MHz is 21.2 watts.

12.9.3. Command Listing

You can perform this example harmonic analysis using the ANSYS commands shown below. Text prefaced by an exclamation point (!) is a comment.

Besides commands to perform this analysis using the [PLANE230](#) electric elements, this command listing includes the commands to do the same analysis using [PLANE121](#) electrostatic elements.

```

/batch,list
/title, Harmonic response of a lossy capacitor
/com,
/com, Problem parameters:
a=9.e-2                      ! radius, m
d=0.1e-2                      ! thickness, m
epsr=1143                      ! relative permittivity
tand=0.0105                     ! loss tangent
Vo=50                          ! voltage amplitude, v
f1=0                            ! begin frequency, Hz
f2=1.e6                         ! end frequency, Hz
eps0=8.854e-12                  ! free space permittivity, F/m
Pi=acos(-1)
C=epsr*eps0*Pi*a**2/d          ! capacitance, F
P2d=Pi*f2*Vo**2*C*tand         ! power dissipation at freq. f2, Watt

/nopr
/PREP7
et,1,PLANE230,,,1                ! axisymmetric electric element
emunit,epzro,eps0                 ! specify free-space permittivity
mp,perx,1,epsr                   ! electric material properties
mp,lsst,1,tand

rect,,a,,d                        ! model and mesh

```

```

esize,d/2
amesh,1

! Boundary conditions and loads
nsel,s,loc,y,0
cp,1,volt,all           ! define bottom electrode
*get,n_grd,node,0,num,min ! get master node on bottom electrode
nsel,s,loc,y,d
cp,2,volt,all           ! top electrode
*get,n_load,node,0,num,min ! get master node on top electrode
nsel,all
d,n_grd,volt,0          ! ground bottom electrode
d,n_load,volt,Vo        ! apply voltage load to top electrode
fini

/solu
antype,harm             ! harmonic analysis
harfrq,f1,f2             ! frequency range
nsubs,10                 ! number of substeps
outres,all,all            ! write all solution items to the result file
kbc,1                     ! stepped load
solve
fini

/post1
/com,Calculate power dissipation at frequency = %f2%, Hz
set,last                  ! read last data set
etab,jh,jheat              ! fill etable with Joule heat rates per unit volume
etab,vol,volu               ! fill etable with element volumes
smult,dpower,jh,vol         ! fill etable with element Joule heat rates
ssum                       ! sum element Joule heat rates
/com,Expected power dissipation = %P2d%, Watt
fini

/post26
rfor,2,n_load,amps         ! reaction current I
prod,3,2,,,Y_ANSYS,,,1/Vo ! Y_ansys = I/V
prod,4,1,,,,,,2*Pi*C      ! 2*Pi*f*C
cfact,tand,,,1
add,5,4,4,,Y_TARGET        ! Y_target = 2*Pi*f*C*(tand+j)
prcplx,1
prvar,Y_ANSYS,Y_TARGET
fini

/com,
/com, *** Perform same frequency sweep using electrostatic elements
/com,
/PREP7
et,1,PLANE121,,,1          ! axisymmetric electrostatic element
fini

/solu
antype,harm             ! harmonic analysis
harfrq,f1,f2             ! frequency range
nsubs,10                 ! number of substeps
outres,all,all            ! write all solution items to the result file
kbc,1                     ! stepped load
solve
fini

/post1
/com,Calculate power dissipation at frequency = %f2%, Hz
set,last                  ! read last dataset
etab,jh,jheat              ! fill etable with Joule heat rates per unit volume

```

```
etab,vol,volu          ! fill etable with element volumes
smult,dpower,jh,vol    ! fill etable with element Joule heat rates
ssum                   ! summ up element Joule heat rates
/com,Expected power dissipation = %P2d%, Watt
fini

/post26
rfor,2,n_load,chrg      ! reaction charge Q
cfact,0,2*Pi
prod,3,1,2,,Y_ANSYS,,,1/Vo   ! Y_ansys = j*2*Pi*Q/Vo
prod,4,1,,,,,,2*Pi*C       ! 2*Pi*f*C
cfact,tand,,,1
add,5,4,4,,Y_TARGET        ! Y_target = 2*Pi*f*C*(tand+j)
prcplx,1
prvar,Y_ANSYS,Y_TARGET
fini
```

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

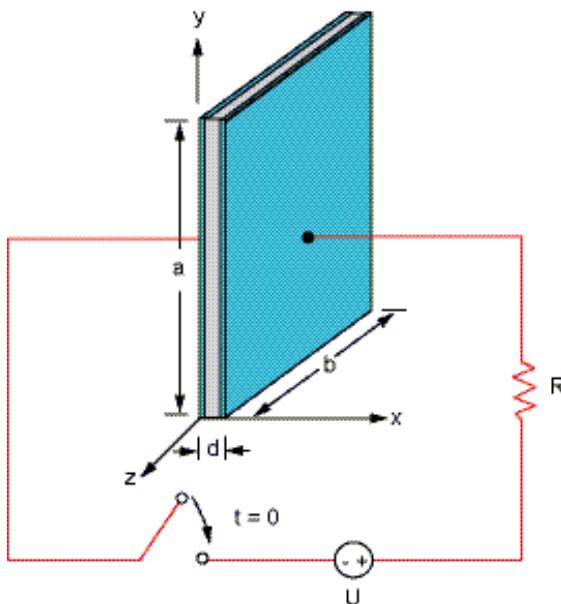
12.10. Sample Transient Quasistatic Electric Analysis

[www.kxcad.net Home](http://www.kxcad.net) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

12.10.1. Problem Description

A parallel plate capacitor filled with a lossy dielectric, characterized by relative permittivity ϵ_r and resistivity ρ , is connected, in series with a resistor R , to a source of constant voltage U . The switch closes at time $t = 0$. Find the electric field and current density distributions in the capacitor as functions of time.

Figure 12.9 Parallel Plate Capacitor Connected to a Voltage Source



The capacitor and circuit parameters are:

Length $a = 3$ cm

Width $b = 1$ cm

Thickness $d = 0.5$ cm

Loss tangent $\tan\delta = 0.0105$

Resistivity $= 150 \Omega\text{m}$

Resistance $R = 1 \text{ k}\Omega$

Voltage $U = 12$ volts

The lossy capacitor is modeled with [SOLID231](#) electric elements. Electrodes are defined by coupling VOLT degrees of freedom on the major surfaces of the capacitor. Lumped circuit components are modeled using [CIRCU124](#) elements and they are connected to the capacitor via the master nodes. A transient analysis is performed to determine the electric field variation in the capacitor with time. Computed results for a selected element are compared in /POST26 to the values derived from the following analytical expression for the electric field:

$$E(t) = E_s \{1 - \exp(-t/\tau)\}$$

where:

E_s = steady state electric field, Volts/m

S = capacitor plate area, m^2

τ = time constant, seconds

and they are defined by:

$$E_s = \frac{U}{\frac{SR}{\rho} + d}$$

$$S = ab$$

$$\tau = \frac{\epsilon_r \epsilon_0 S R}{\frac{SR}{\rho} + d}$$

12.10.2. Results

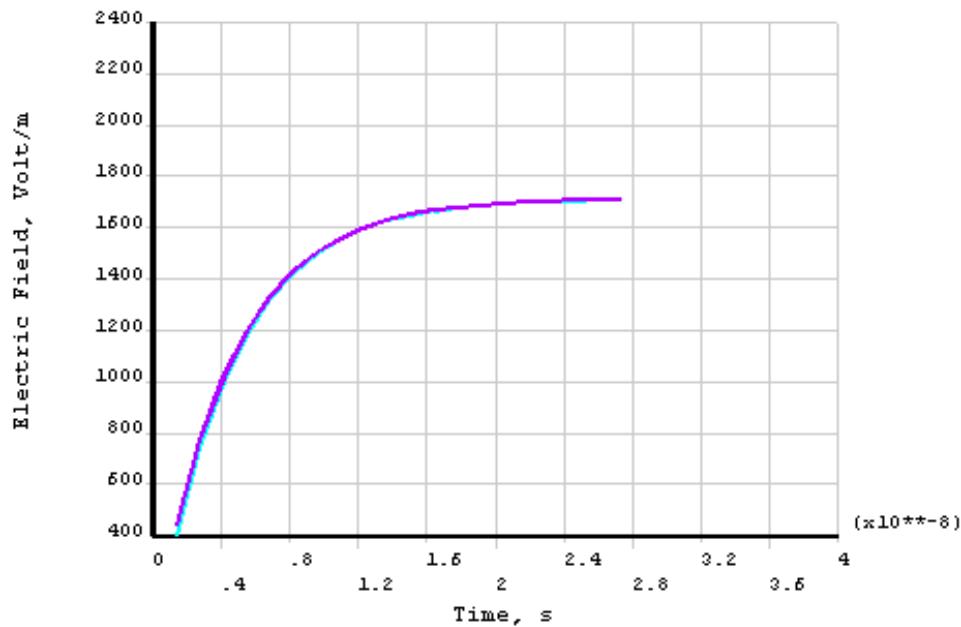
The following figures display the electric field and current density distributions as functions of time.

Figure 12.10 Computed and Target Electric Fields

1
POST26

EF_ANS

EF_TAR

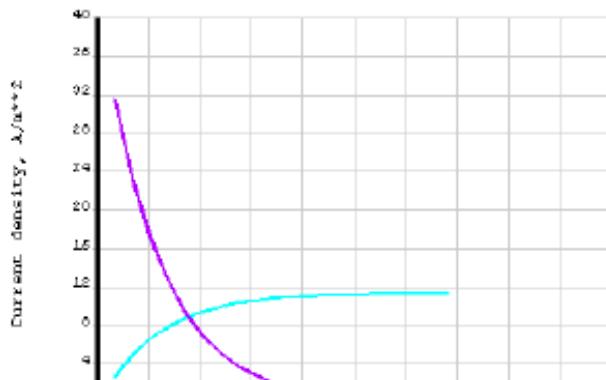


Transient effects in a lossy capacitor

Figure 12.11 Current Density (Conduction, Displacement, and Total)1
POST26

JC_ANS

JD_ANS



12.10.3. Command Listing

You can perform this example transient analysis using the ANSYS commands shown below. Text prefaced by an exclamation point (!) is a comment.

Besides /POST26 commands to display the electric field and current density, this command listing includes the commands to print electric field, current density, electric flux density, and Joule heat rate. The current density and Joule heat rate output quantities are included to illustrate the coupling to magnetic and thermal analyses using companion magnetic and thermal elements, respectively.

```

/batch,list
/title, Transient effects in a lossy capacitor
/com,
/com, Problem parameters:
a=3.e-2           ! length, m
b=1.e-2           ! width, m
S=a*b             ! area, m**2
d=0.5e-2          ! thickness, m
epsr=12            ! relative permittivity of dielectric
eps0=8.854e-12    ! free space permittivity, F/m
rho=150            ! resistivity of dielectric, Ohm*m,
R=1000             ! resistance, Ohm
U=12               ! voltage, V
Es=U/ (S*R/rho+d) ! steady-state electric field, V/m
tau=epsr*eps0*S*R/(S*R/rho+d) ! time constant, s

/nopr
/PREP7
emunit,epzro,eps0 ! Specify free-space permittivity

! Element attributes
et,1,CIRCU124     ! Resistor
et,2,CIRCU124,4   ! Voltage source
et,3,SOLID231      ! 20-node brick electric solid
r,1,R              ! Real constants for circuit elements
r,2,U
mp,rsvx,1,rho      ! Electric properties
mp,perx,1,epsr

! Modeling and meshing
type,3
mat,1
block,,d,,a,,b
esize,d/2
vmesh,1

nsel,s,loc,x,0
cp,1,volt,all      ! Couple nodes to model right electrode
*get,n1,node,0,num,min ! Get master node on right electrode
nsel,s,loc,x,d
cp,2,volt,all      ! Couple nodes to model left electrode
*get,n2,node,0,num,min ! Get master node on left electrode
nsel,all

```

```

! Circuit mesh
*get,nmax,node,0,num,max
n3=nmax+1
n4=n3+1
n,n3,a/2,-a
n,n4,a/2,-a
type,1           ! resistor
real,1
e,n1,n3
type,2           ! voltage source
real,2
e,n3,n2,n4
d,n2,volt,0      ! ground node
fini

/solu
antype,transient    ! transient analysis
t1=6*tau
time,t1             ! set analysis time
deltim,t1/100        ! set time step
outres,all,5          ! write results at every 5th substep
ic,all,volt,0          ! initial conditions for VOLT
solve
fini

! Select nodes and elements for post-processing
n_post=node(d/2,a/2,b/2)
nsel,s,node,,n_post
esln,s
*get,e_post,elem,0,num,min
allsel

/com, *** Results verification
/post26
numvar,20

! Electric field
esol,2,e_post,,EF,x,EF_ANS
exp,3,1,,,,,-1/tau,-1           ! -exp(-t/tau)
filldata,4,,,1                  ! 1
add,5,4,3,,EF_TAR,,,Es,Es      ! E=Es*(1-exp(-t/tau))

! Conduction current
esol,6,e_post,,JC,x,JC_ANS
prod,7,5,,,JC_TAR,,,1/rho       ! Jc=E/rho

! Electric flux density
esol,8,e_post,,NMISC,1,D_ANS
prod,9,5,,,D_TAR,,,epsr*eps0    ! D=epsr*eps0*E

! Displacement and total (displacement+conduction) currents
esol,10,e_post,,JS,x,JS_ANS

```

```

add,11,10,6,,JD_ANS,,,,-1           ! Jd=Js-Jc
deriv,12,9,1,,JD_TAR                ! Jd=dD/dt
add,13,7,12,,JS_TAR

! Joule heat generation rate per unit volume
esol,14,e_post,,JHEAT,,HGEN_ANS
prod,15,5,7,,HGEN_TAR! Jheat=E*Jc

prvar,EF_ANS,EF_TAR,D_ANS,D_TAR
prvar,JC_ANS,JC_TAR,JD_ANS,JD_TAR,JS_ANS,JS_TAR
prvar,HGEN_ANS,HGEN_TAR

/axlab,x, Time, s
/axlab,y, Electric Field, Volt/m
plvar,EF_ANS,EF_TAR          ! Plot computed and expected EF
/axlab,y, Current density, A/m**2
plvar,JC_ANS,JD_ANS,JS_ANS    ! Plot computed currents
fini

/com, *** Coupling to thermal analysis
/PREP7
et,1,0
et,2,0
et,3,90           ! companion thermal element
esel,s,elem,,e_post
ldread,hgen,,5,,,rth ! read heat generation from the 5th substep
bfelist,all,hgen
ldread,hgen,,,,rth   ! read heat generation from the last substep
bfelist,all,hgen
esel,all
fini

/com, *** Coupling to magnetic analysis
/PREP7
et,3,117          ! companion magnetic element
esel,s,elem,,e_post
ldread,js,,5,,,rth ! read current from the 5th substep
bfelist,all,js
ldread,js,,,,rth    ! read current from the last substep
bfelist,all,js
esel,all
fini

```

[Low-Frequency Guide](#) | [Chapter 12. Electric Field Analysis](#) |

12.11. Where to Find Current Conduction Analysis Examples

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

Another ANSYS, Inc., publication, the [*Verification Manual*](#), contains several examples of current conduction analysis:

[VM117](#) - Electric Current Flowing in a Network

[VM170](#) - Magnetic Field from a Square Current Loop

[VM173](#) - Centerline Temperature of an Electrical Wire

[Low-Frequency Guide](#) |

Chapter 13. Electrostatic Field Analysis (h-Method)

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

Electrostatic field analysis determines the electric field and electric scalar potential (voltage) distribution caused by charge distributions or applied potential. You can apply two types of loads in this analysis: voltage and charge densities.

An electrostatic analysis is assumed to be linear. The electric field is proportional to the applied voltage.

Two electrostatic analyses methods are available: h-Method and p-Method. The traditional h-Method is covered in this chapter and the p-Method is covered in "[p-Method Electrostatic Analysis](#)".

Electrostatic contact is also available in ANSYS. See [Modeling Electric Contact](#) in the [Contact Technology Guide](#) for details.

The following h-Method electrostatic field analysis topics are available:

- [Elements Used in h-Method Electrostatic Analysis](#)
 - [Steps in an h-Method Electrostatic Analysis](#)
 - [Extracting Capacitance from Multi-conductor Systems](#)
 - [Trefftz Method for Open Boundary Representation](#)
 - [Doing an Example h-Method Electrostatic Analysis \(GUI Method\)](#)
 - [Doing an Electrostatic Analysis \(Command Method\)](#)
 - [Doing an Example Capacitance Calculation \(Command Method\)](#)
 - [Doing an Electrostatic Analysis Using Trefftz Method \(Command Method\)](#)
 - [Doing an Electrostatic Analysis Using Trefftz Method \(GUI Method\)](#)
-
-

[Low-Frequency Guide](#) | [Chapter 13. Electrostatic Field Analysis \(h-Method\)](#) |

13.1. Elements Used in h-Method Electrostatic Analysis

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

Electrostatic analyses use the following ANSYS elements:

Table 13.1 2-D Solid Elements

| Element | Dimens. | Shape or Characteristic | DOFs | Notes |
|--------------------------|---------|----------------------------|----------------------|---|
| PLANE121 | 2-D | Quadrilateral, eight nodes | Voltage at each node | Supported for cyclic symmetry (periodic) analyses, except for Trefftz method or coupled thermo-electric with VOLT and TEMP. |

Table 13.2 3-D Solid Elements

| Element | Dimens. | Shape or Characteristic | DOFs | Notes |
|--------------------------|---------|-------------------------|----------------------|---|
| SOLID122 | 3-D | Brick, 20 nodes | Voltage at each node | Supported for cyclic symmetry (periodic) analyses, except for Trefftz method or coupled thermo-electric with VOLT and TEMP. |
| SOLID123 | 3-D | Tetrahedral, 10 node | Voltage at each node | Supported for cyclic symmetry (periodic) analyses, except for Trefftz method or coupled thermo-electric with VOLT and TEMP. |

Table 13.3 Specialty Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|--------------------------|---------------------------------|---|--|
| MATRIX50 | None (this is a super- element) | Depends on the elements that make up this element | Depends on the included element types |
| INFIN110 | 2-D | Four or eight nodes | One per node; this can be a magnetic vector potential, temperature, or electric potential |
| INFIN111 | 3-D | Hexalateral, 8 or 20 nodes | A X, A Y, A Z magnetic vector potential, temperature, electric potential, or magnetic scalar potential |
| INFIN9 | 2-D | Planar, unbounded, two nodes | A Z magnetic vector potential, temperature |
| INFIN47 | 3-D | Quadrilateral, four nodes or | A Z magnetic vector potential, temperature |

triangle, three nodes

[Low-Frequency Guide](#) | [Chapter 13. Electrostatic Field Analysis \(h-Method\)](#) |

13.2. Steps in an h-Method Electrostatic Analysis

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

The procedure for doing an electrostatic analysis consists of three main steps:

1. Build the model.
2. Apply loads and obtain the solution.
3. Review the results.

The next few topics discuss what you must do to perform these steps. First, the text presents a general description of the tasks required to complete each step. An example follows, based on an analysis of electrostatic forces between charge spheres. The example walks you through doing the analysis by choosing items from ANSYS' GUI menus, then shows you how to perform the same analysis using ANSYS commands.

13.2.1. Building the Model

To build the model, you start by specifying the jobname and a title for your analysis, using the following commands or GUI paths:

Command(s): [/FILNAME](#)
[/TITLE](#)

GUI: Utility Menu> File> Change Jobname
 Utility Menu> File> Change Title

If you are using the ANSYS GUI, the next step is to set preferences for an electric analysis: **Main Menu> Preferences> Electromagnetics: Electric**

You *must* set the preference to Electric to ensure that the elements needed for your analysis will be available. (The ANSYS GUI filters element types based on the preference you choose.)

Once you have set the Electric preference, use the ANSYS preprocessor (PREP7) to define the element types, the material properties, and the model geometry. These tasks are common to most analyses. The [Modeling and Meshing Guide](#) explains them in detail.

For an electrostatic analysis, you must define the permittivity (PERX) material property. It can be temperature dependent, as well as isotropic or orthotropic.

In ANSYS, you must make sure that you use a consistent system of units for all the data you enter. See the [EMUNIT](#) command in the [Commands Reference](#) for additional information regarding appropriate settings of

free-space permeability and permittivity.

For micro-electromechanical systems (MEMS), it is best to set up problems in more convenient units since components may only be a few microns in size. For convenience, see [Table 13.4: "Electrical Conversion Factors for MKS to μMKSv"](#) and [Table 13.5: "Electrical Conversion Factors for MKS to μMSVfA"](#).

Table 13.4 Electrical Conversion Factors for MKS to μMKSv

| Electrical Parameter | MKS Unit | Dimension | Multiply by This Number | To Obtain μMKSv Unit | Dimension |
|-----------------------|--------------------|---|-------------------------|----------------------|---|
| Voltage | V | (kg)(m) ² /(A)(s) ³ | 1 | V | (kg)(μm) ² /(pA)(s) ³ |
| Current | A | A | 10 ¹² | pA | pA |
| Charge | C | (A)(s) | 10 ¹² | pC | (pA)(s) |
| Conductivity | S/m | (A) ² (s) ³ /(kg)(m) ³ | 10 ⁶ | pS/μm | (pA) ² (s) ³ /(kg)(μm) ³ |
| Resistivity | Ωm | (kg)(m) ³ /(A) ² (s) ³ | 10 ⁻⁶ | T Ωμm | (kg)(μm) ³ /(pA) ² (s) ³ |
| Permittivity [1] | F/m | (A) ² (s) ⁴ /(kg)(m) ³ | 10 ⁶ | pF/μm | (pA) ² (s) ² /(kg)(μm) ³ |
| Energy | J | (kg)(m) ² /(s) ² | 10 ¹² | pJ | (kg)(μm) ² /(s) ² |
| Capacitance | F | (A) ² (s) ⁴ /(kg)(m) ² | 10 ¹² | pF | (pA) ² (s) ⁴ /(kg)(μm) ² |
| Electric Field | V/m | (kg)(m)/(s) ³ (A) | 10 ⁻⁶ | V/μm | (kg)(μm)/(s) ³ (pA) |
| Electric Flux Density | C/(m) ² | (A)(s)/(m) ² | 1 | pC/(μm) ² | (pA)(s)/(μm) ² |

- Free-space permittivity is equal to 8.854×10^{-6} pF/μm.

Table 13.5 Electrical Conversion Factors for MKS to μMSVfA

| Electrical Parameter | MKS Unit | Dimension | Multiply by This Number | To Obtain μMSVfA Unit | Dimension |
|-----------------------|--------------------|---|-------------------------|-----------------------|--|
| Voltage | V | (kg)(m) ² /(A)(s) ³ | 1 | V | (g)(μm) ² /(fA)(s) ³ |
| Current | A | A | 10 ¹⁵ | fA | fA |
| Charge | C | (A)(s) | 10 ¹⁵ | fC | (fA)(s) |
| Conductivity | S/m | (A) ² (s) ³ /(kg)(m) ³ | 10 ⁹ | fS/μm | (fA) ² (s) ³ /(g)(μm) ³ |
| Resistivity | Ωm | (Kg)(m) ³ /(A) ² (s) ³ | 10 ⁻⁹ | - | (g)(μm) ³ /(fA) ² (s) ³ |
| Permittivity [1] | F/m | (A) ² (s) ⁴ /(kg)(m) ³ | 10 ⁹ | fF/μm | (fA) ² (s) ² /(g)(μm) ³ |
| Energy | J | (kg)(m) ² /(s) ² | 10 ¹⁵ | fJ | (g)(μm) ² /(s) ² |
| Capacitance | F | (A) ² (s) ⁴ /(kg)(m) ² | 10 ¹⁵ | fF | (fA) ² (s) ⁴ /(g)(μm) ² |
| Electric Field | V/m | (kg)(m)/(s) ³ (A) | 10 ⁻⁶ | V/μm | (g)(μm)/(s) ³ (fA) |
| Electric Flux Density | C/(m) ² | (A)(s)/(m) ² | 10 ³ | fC/(μm) ² | (fA)(s)/(μm) ² |

- Free-space permittivity is equal to 8.854×10^{-3} fF/μm.

13.2.2. Applying Loads and Obtaining a Solution

In this step, you define the analysis type and options, apply loads to the model, specify load step options, and initiate the finite element solution. The next few topics explain how to perform these tasks.

13.2.2.1. Entering the SOLUTION Processor

To enter the SOLUTION processor, use either of the following:

Command(s): [/SOLU](#)

GUI: Main Menu> Solution

13.2.2.2. Defining the Analysis Type

To specify the analysis type, do either of the following:

- In the GUI, choose menu path **Main Menu> Solution> Analysis Type> New Analysis** and choose a Static analysis.
- If this is a new analysis, issue the command [ANTYPE,STATIC,NEW](#).
- If you want to restart a previous analysis (for example, to specify additional loads), issue the command [ANTYPE,STATIC,REST](#). You can restart an analysis only if you previously completed an electrostatic analysis, and the files Jobname.EMAT, Jobname.ESAV, and Jobname.DB from the previous run are available.

13.2.2.3. Defining Analysis Options

Next, you define which solver you want to use. You can select the sparse solver (default), Preconditioned Conjugate Gradient (PCG) solver, PCG out-of-memory solver, Jacobi Conjugate Gradient (JCG) solver, Incomplete Cholesky Conjugate Gradient (ICCG) solver, or JCG out-of-memory solver.

To select an equation solver, use either of the following:

Command(s): [EOSLV](#)

GUI: Main Menu> Solution> Analysis Type> Analysis Options

If you choose either the JCG solver or the PCG solver, you can also specify a solver tolerance value. This value defaults to 1.0E-8.

13.2.2.4. Apply Boundary Conditions

These loads specify flux-parallel, flux-normal, far-field, and periodic boundary conditions, as well as an imposed external magnetic field. The following table shows the value of VOLT required for each type of boundary condition:

| Boundary Condition | Value of VOLT |
|--------------------|--|
| Flux-parallel | None required (naturally occurring). |
| Flux-normal | Specify a constant value of VOLT using the <u>D</u> command (Main Menu> Solution> Define Loads> Apply> Electric> Boundary> Voltage> J-Normal> On Nodes (On |

| Lines or On Areas). | |
|----------------------------|--|
| Far-field | For 2-D analysis, use INFIN9 (planar analyses only) or INFIN110 elements. For 3-D analysis, use the Trefftz formulation or INFIN47 or INFIN111 elements. |
| Imposed external field | Apply nonzero values of VOLT. Use Main Menu> Solution> Define Loads> Apply> Electric> Boundary> Voltage> On Keypoints (On Nodes, On Lines or On Areas) . |

Flux-parallel boundary conditions force the flux to flow parallel to a surface, while flux-normal boundary conditions force the flux to flow normal to a surface. You do not need to specify far-field zero boundary conditions if you use the Trefftz formulation or far-field elements to represent the "infinite" boundary of the model. For an imposed external field, specify the appropriate nonzero value of VOLT.

13.2.2.5. Applying Loads

You can apply loads to an electrostatic analysis either on the solid model (keypoints, lines, and areas) or on the finite element model (nodes and elements). You can specify several types of loads:

13.2.2.5.1. Voltage (VOLT)

These loads are DOF constraints that you usually specify at model boundaries to apply a known voltage. To apply voltage, use one of the following:

Command(s): [**D**](#)

GUI: **Main Menu> Solution> Define Loads> Apply> Electric> Boundary> Voltage**

13.2.2.5.2. Charges (CHRG)

These are concentrated nodal force loads. To apply them, use the following command or GUI path:

Command(s): [**F**](#)

GUI: **Main Menu> Solution> Define Loads> Apply> Electric> Excitation> Charge> On Nodes**

13.2.2.5.3. Surface Charge Densities (CHRGS)

These are surface loads you can apply at nodes or elements. To apply them, use the following command or GUI path:

Command(s): [**SF**](#)

GUI: **Main Menu> Solution> Define Loads> Apply> Electric> Excitation> Surf Chrg Den**

13.2.2.5.4. Infinite Surface Flags (INF)

[Low-Frequency Guide](#) | [Chapter 13. Electrostatic Field Analysis \(h-Method\)](#) |

13.3. Extracting Capacitance from Multi-conductor Systems

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

A key parameter from an electrostatic solution is capacitance. For multiple conductor systems, this involves extracting self and mutual capacitance terms so that equivalent circuit lumped capacitors can be defined for use in circuit simulators. The **CMATRIX** command macro has been developed to extract self and mutual capacitance terms for multiple conductor systems. See the [Theory Reference for ANSYS and ANSYS Workbench](#) for more details.

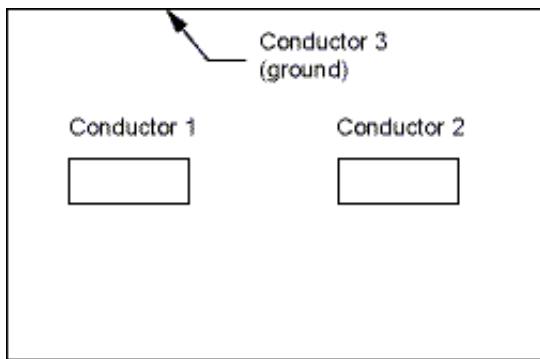
13.3.1. Ground Capacitances and Lumped Capacitances

Finite element simulation can readily compute and extract a "Ground" capacitance matrix of capacitance values that relate the charge on one conductor with the conductor's voltage drop (to ground). [Figure 13.2: "Three Conductor System"](#) illustrates a three-conductor system (one conductor is ground). The following two equations relate charges on electrodes 1 and 2, Q_1 and Q_2 , with the voltage drops for the electrodes, U_1 and U_2 :

$$Q_1 = (C_g)_{11} (U_1) + (C_g)_{12} (U_2)$$

$$Q_2 = (C_g)_{12} (U_1) + (C_g)_{22} (U_2)$$

Figure 13.2 Three Conductor System

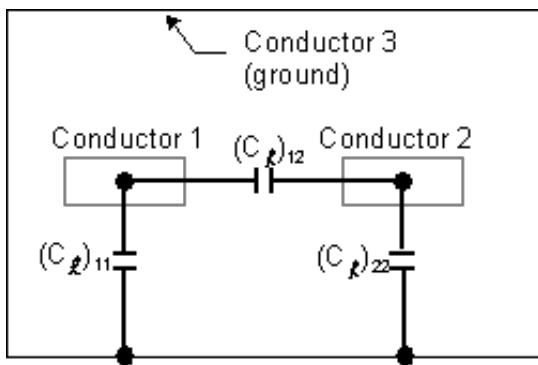


where C_g represents a matrix of capacitances referred to as "ground capacitances". These ground capacitances do not represent lumped capacitances typically used in a circuit simulator because they do not relate the capacitances between conductors. However, the **CMATRIX** command macro can convert the ground capacitance matrix to a lumped capacitor matrix which is suitable for use in circuit simulators. [Figure 13.3: "Lumped Capacitor Equivalence of Three Conductor System"](#) illustrates the lumped capacitances between the conductors. The following two equations then relate the charges with the voltage drops:

$$Q_1 = (C_l)_{11} (U_1) + (C_l)_{12} (U_1 - U_2)$$

$$Q_2 = (C_l)_{12} (U_1 - U_2) + (C_l)_{22} (U_2)$$

Figure 13.3 Lumped Capacitor Equivalence of Three Conductor System



where C_l represents a matrix of capacitances referred to as “lumped capacitances”.

13.3.2. Procedure

The **CMATRIX** command macro will perform multiple simulations and extract both the ground capacitance matrix values and the lumped capacitance matrix values. To prepare for **CMATRIX**, you must group the conductor nodes into node components. Do not apply any loads to the model (voltages, charge, charge density, etc). The component name applied to the conductor nodes must contain a common prefix, followed by a numerical suffix progressing from 1 to the highest numbered conductor in the system. The last numbered conductor in the system must be the ground conductor (the conductor whose potential is assumed to be zero). The procedure for using **CMATRIX** is as follows:

1. Build and mesh the solid model with electrostatic elements. Conductors are assumed to be perfect conductors and hence do not require a finite element mesh within the conductor domain. Only the surrounding dielectric regions and air regions require a mesh. The resulting nodes on the boundary of the conductors represent the nodes that will be grouped into node components.
2. Select the nodes on the surface of the each conductor and group them into node components.

Command(s): CM

GUI: Utility Menu> Select> Comp/Assembly> Create Component

Share a command prefix for the component names, and use a numerical value sequencing from 1 to the highest numbered conductor. For example, in [Figure 13.3: "Lumped Capacitor Equivalence of Three Conductor System"](#), three node components would be defined for each set of conductor nodes. Using a prefix "cond", the node component names would be "cond1", "cond2", and "cond3". The last component, "cond3" would be the nodes representing the ground.

3. Enter the SOLUTION processor, using either of the following:

Command(s): SOLU

GUI: Main Menu> Solution

4. Select an equation solver (JCG recommended), using either of the following:

Command(s): [EQSLV](#)**GUI:** Main Menu> Solution> Analysis Type> Analysis Options

5. Invoke the **CMATRIX** macro, using one of the following:

Command(s): [CMATRIX](#)**GUI:** Main Menu> Solution> Solve> Electromagnet> Static Analysis> Capac Matrix

The **CMATRIX** command macro requires the following input:

- A symmetry factor (*SYMFAC*). If there is no symmetry in the model, the symmetry factor is 1 (default). If you wish to model only a portion of the model by taking advantage of symmetry, use the symmetry factor as a multiplier to obtain the correct capacitance.
- The node component prefix name (*Condname*). This is the prefix of the node component name's used to define the conductor node components. In the above example, the prefix name is "cond". The command macro requires that you put single quotes around the prefix name when entering the character string. Thus, the input for this example would be 'cond'. In the GUI, the single quotes are automatically handled by the program.
- The number of conductor node components (*NUMCOND*). Insert the total number of conductor node components. In the above example you would use "3".
- Enter the Ground Key option (*GRNDKEY*). If your model does not contain an open boundary then the highest numbered node component represents ground. In this case, no special treatment is needed and you would set the ground key to zero (default). If your model contains an open boundary (modeled with infinite elements, or a Trefftz domain) and the far-field is not considered as a conductor, then you would set the ground key to zero (default). In some situations it is necessary to consider the far-field (infinity) as the ground conductor (for example, a single charged sphere in air requires the infinity location as ground in order to preserve a charge balance). When using the [INFIN111](#) element or a Trefftz domain to represent a far-field ground, set the ground key to "1".
- Enter a name for the stored matrix of capacitance values (*Capname*). The command macro stores the computed ground and lumped matrix values in a 3-D array vector where the "i" and "j" columns represent the conductor indices, the "k" column indicates ground (k = 1) or lumped (k = 2) terms. The default name is CMATRIX. For example, the command macro stores the ground terms in CMATRIX(i,j,1) and the lumped terms in CMATRIX(i,j,2). The command macro also creates a text file containing the matrix values and stores it in a file with the stored matrix name and a .TXT extension.

Do not apply inhomogeneous loads before using the **CMATRIX** command. Inhomogeneous loads are those created by:

- Degree of freedom commands (**D**, **DA**, etc.) specifying nonzero degree of freedom values on nodes or solid model entities
- Force commands (**F**, **BF**, **BFE**, **BFA**, etc.) specifying nonzero force values on nodes, elements, or solid

model entities

- Any **CE** command with a nonzero constant term

CMATRIX executes a series of solutions to compute self and mutual capacitance between conductors. The solutions, which are stored in the results file, are available for postprocessing, if desired. At the end of the execution, the command macro presents a summary table.

If infinite elements (**INFIN110** and **INFIN111**) share a common boundary with a conductor (such as a ground plane), you can consider the ground plane and infinite boundary as a single conductor (group only the ground plane nodes into a component).

[Figure 13.4: "Modeling Scenarios"](#) illustrates several modeling scenarios for open and closed domain models along with the appropriate settings for

[Low-Frequency Guide](#) | [Chapter 13. Electrostatic Field Analysis \(h-Method\)](#) |

13.4. Trefftz Method for Open Boundary Representation

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

In conjunction with the Infinite Elements ([INFIN110](#) and [INFIN111](#)) for modeling the open domain of a field problem, another method may be chosen that utilizes a hybrid finite element - Trefftz method (hereafter referred to as the Trefftz method). The Trefftz method bears the name of the founder of boundary element techniques. The Trefftz method combines the efficiency of boundary techniques in open domain treatment with a finite element-like positive definite stiffness matrix. It allows treatment of complex surface geometry, even with high aspect ratios. It offers an easy to generate Trefftz-complete function system. The result is an easy to use, accurate method for handling open boundary domains in electrostatics. For information on the Trefftz Method theory, see the [Theory Reference for ANSYS and ANSYS Workbench](#). For an example problem, see [Doing an Electrostatic Analysis Using Trefftz Method \(Command Method\)](#) in this manual.

13.4.1. Overview

The Trefftz method involves creating a Trefftz domain. The Trefftz domain consists of the following:

- A set of Trefftz source nodes located within the finite element domain, but not attached to the finite element model.
- A flagged exterior surface of the finite element region.
- A substructure matrix created from the Trefftz source nodes and the flagged exterior finite element surfaces.
- A superelement defined from the substructure.
- A set of constraint equations generated in conjunction with the substructure.

The Trefftz method has a number of positive features and certain advantages and disadvantages when compared to infinite elements.

The Trefftz method offers the following positive features:

- The formulation leads to symmetric matrices.
- The formulation has no theoretical limitations to open boundary treatment.
- The method does not involve a singular integral.
- The number of unknowns is minimal. (20-100 unknowns give reasonable results.)

- The method may be applied to high aspect ratio boundaries.
- The method allows flexible generation of Green's functions.
- The Trefftz region can establish relations between isolated FEM domains.

The Trefftz method has the following advantages over infinite elements:

- It is generally more accurate.
- It does not require modeling and meshing an infinite element region.
- It can be used on high aspect ratio finite element regions with good accuracy.
- It does not require the finite element region to extend as much beyond the modeled region of the device as is required with infinite elements.

The Trefftz method has the following disadvantages compared to infinite elements:

- It cannot be used if employing symmetry in the model.
- It is available only for 3-D analysis.
- It requires that the finite element mesh at the exterior of the model be tetrahedral elements only.
- It requires the definition of Trefftz source nodes within the finite element domain and the generation of a substructure and constraint equations. (This process, however, has been highly automated.)

The Trefftz method has the following limitations:

- 10,000 is the maximum number of Trefftz nodes.
- 10,000,000 is the highest allowable node number.
- 5,000,000 is the maximum number of nodes allowed on the exterior surface.
- 5,000,000 is the maximum number of elements (facets) allowed on the exterior surface.

The Trefftz formulation assumes a zero potential at infinity. Accordingly, you must exercise care when applying the Trefftz method to a multi-electrode system of differing applied potentials. However, for capacitance computations using a Trefftz domain, the **CMATRIX** command properly accounts for a zero or floating potential at infinity.

13.4.2. Procedure

To create a Trefftz domain in a 3-D electrostatic analysis, you perform the following tasks as illustrated in [Figure 13.5: "Defining a Trefftz Domain"](#):

1. Create a finite element model of the electrostatic domain (including conductors, dielectrics, and surrounding air). Apply all necessary boundary conditions to the finite element model (voltages, charge,

charge density, etc.).

2. Flag the exterior surface of the finite element region as an infinite surface. To apply the infinite surface flag (INF label), use one of the following:

Command(s): SF,
SFA,
SFE

GUI: Main Menu> Preprocessor> Loads> Define Loads> Apply> Electric> Flag> Infinite Surf> On Nodes
Main Menu> Preprocessor> Loads> Define Loads> Apply> Electric> Flag> Infinite Surf> On Areas
Main Menu> Preprocessor> Trefftz Domain> Infinite Surf> On Areas

3. Create the Trefftz source nodes. The Trefftz source nodes act as the unknowns of the Trefftz domain. These unknowns represent source charges used in the Trefftz method. The source charges are computed and stored for the Trefftz nodes using the CURR degree of freedom.

As shown in [Figure 13.5: "Defining a Trefftz Domain"](#), Step 3, you should locate the Trefftz source nodes between the modeled device and the exterior of the finite element region. The Trefftz method will be more accurate when the Trefftz nodes are closer to the modeled device than the exterior of the finite element model. Also, a greater accuracy will be obtained the further the Trefftz sources are from the surface of the finite element model. For example, as shown for the x direction, the Trefftz nodes should just encompass the modeled device ($b/c > 1$), and the finite element exterior boundary should be located a greater distance away ($a/b > 2$). You should apply similar rules for the y and z directions. You should not place Trefftz source nodes close to, or on the surface of the finite element domain. This may lead to a near-singular solution and produce inaccurate results.

You can easily create the Trefftz nodes by defining a simple solid model object (block, sphere, cylinder or Boolean union of them) that encompasses the modeled device, but is interior to the exterior finite element region as shown in [Figure 13.5: "Defining a Trefftz Domain"](#).

Once the simple solid model object is defined, you can mesh the simple solid model to create the Trefftz nodes. To do so, use one of the following:

Command(s): TZAMESH
GUI: Main Menu> Preprocessor> Trefftz Domain> Mesh TZ Geometry

The **TZAMESH** command meshes the surface areas of the volume and then clears the non-solution element, leaving only the Trefftz nodes. It groups the Trefftz nodes into a node component called TZ_NOD for future use in the generation of the Trefftz substructure.

The Trefftz method requires only a few source nodes. By default, **TZAMESH** will create two divisions along each edge of the simple solid model entity. For high aspect ratio geometry's, you may elect to mesh the entity according to a prescribed length. Both options are available with the **TZAMESH** command. A larger number of Trefftz nodes tends to provide, but does not guarantee, greater accuracy in the solution. Accuracy also is affected by the number of exterior flagged finite elements and their proximity to the Trefftz sources. Typical problems will not require more than 20 to 100 Trefftz nodes.

If you need to you can delete the Trefftz nodes by issuing the following:

CMSFL,TZ_NOD**Command(s):** NDFLE ALL**CMDELE** TZ_NOD**GUI:** Main Menu> Preprocessor> Trefftz Domain> Delete TZ Nodes

4. Create the Trefftz substructure, superelement, and constraint equations. The Trefftz method uses the flagged exterior surface facets from the finite element model and the Trefftz nodes to create a substructure matrix. This matrix is brought into the model as a single superelement using the MATRIX50 element type. In addition, a set of constraint equations is required to complete the Trefftz domain. The process of creating the substructure and incorporating it into the model as a superelement as well as defining the constraint equations is entirely automated by using the TZEGEN command macro.

To create the substructure and incorporate it into the model as a superelement, use one of the following:

Command(s): TZEGEN**GUI:** Main Menu> Preprocessor> Trefftz Domain> Superelement> Generate TZ

The TZEGEN command also automatically defines the constraint equations.

Once a Trefftz domain is created, you solve a problem using standard solution procedures.

The Trefftz domain should be deleted if any change is made to the finite elements at the surface of the mesh, or if a new Trefftz domain is to be created. Only one Trefftz domain can exist in the model at a time.

To delete the Trefftz superelement, associated constraint equations and all supporting Trefftz files, use one of the following:

Command(s): TZDELE**GUI:** Main Menu> Preprocessor> Trefftz Domain> Superelement> Delete TZ

TZDELE deletes all the Trefftz files created during the superelement generation. This includes the following:

- Jobname.TZN - Trefftz source nodes
- Jobname.TZE - Trefftz surface facets on the finite element boundary
- Jobname.TZX - Surface nodes on the finite element boundary
- Jobname.TZM - Trefftz material file

See Doing an Electrostatic Analysis Using Trefftz Method (Command Method) of this manual for an example problem using the Trefftz methodology.

Figure 13.5 Defining a Trefftz Domain

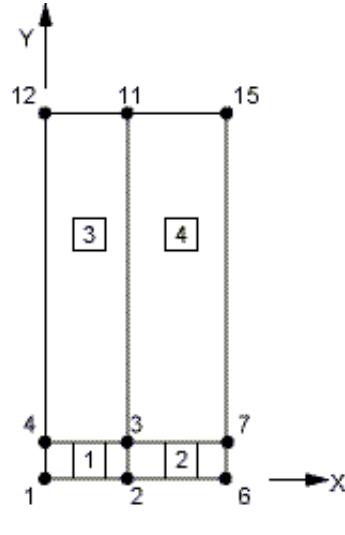
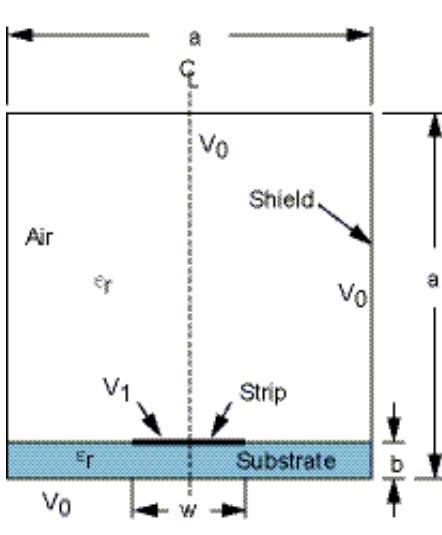
[Low-Frequency Guide](#) | [Chapter 13. Electrostatic Field Analysis \(h-Method\)](#) |

13.5. Doing an Example h-Method Electrostatic Analysis (GUI Method)

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

This section describes how to do an electrostatic analysis of a shielded microstrip transmission line consisting of a substrate, microstrip, and a shield. The strip is at a potential V_1 , and the shield is at a potential V_0 . The object of the problem is to determine the capacitance of the transmission line. See [Doing an Electrostatic Analysis \(Command Method\)](#), to see how to perform the same example analysis by issuing ANSYS commands, either manually during a session or in batch mode.

13.5.1. The Example Described



Material Properties

Air: $\epsilon_r = 1$

Substrate: $\epsilon_r = 10$

Geometric Properties

$a = 10 \text{ cm}$

$b = 1 \text{ cm}$

$w = 1 \text{ cm}$

Loading

$V_1 = 10 \text{ V}$

$V_0 = 1 \text{ V}$

13.5.2. Analysis Assumptions and Modeling Notes

You can calculate the capacitance of the device from electrostatic energy and the applied potential difference as $W_e = 1/2 C (V_1 - V_0)^2$ where W_e is the electrostatic energy and C is the capacitance. To obtain the electrostatic energy, you sum the energies for all the elements in the model in POST1.

Additional postprocessing includes displaying equipotential lines and the electric field as vectors.

13.5.3. Expected Analysis Results

The target results from this example analysis are as follows:

| | Target |
|-------------------|--------|
| Capacitance, pF/m | 178.1 |

Step 1: Begin the Analysis

1. Enter the ANSYS program. To do so, use the procedures described in the [*Operations Guide*](#).
2. Choose menu path **Utility Menu> File> Change Title**. The Change Title dialog box appears.
3. Enter the title "Microstrip transmission line analysis."
4. Click OK.
5. Choose **Main Menu> Preferences**. The Preferences for GUI Filtering dialog box appears.
6. Click **Magnetic-Nodal** and **Electric** on.
7. Click OK.

[**Back To Top**](#)

Step 2: Define Parameters

To do so, follow these steps:

1. Choose **Utility Menu> Parameters> Scalar Parameters**. A dialog box appears.
2. Type in the parameter values shown below, following each entry by pressing ENTER. If you make a mistake, just retype the parameter definition.

V1 = 1.5

V0 = 0.5

3. Click Close to close the dialog box.

[**Back To Top**](#)

Step 3: Define the Element Type

1. Choose **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**. The Element Types dialog box appears.
2. Click Add. The Library of Element Types dialog box appears.
3. Highlight (click on) "Electrostatic" and "2D Quad 121."
4. Click OK. The Element Types dialog box shows element type 1 ([**PLANE121**](#)) selected.

5. Click Close.

[Back To Top](#)

Step 4: Define Material Properties

1. Choose **Main Menu> Preprocessor> Material Props> Material Models**. The Define Material Model Behavior dialog box appears.
2. In the Material Models Available window, double-click on the following options: Electromagnetics, Relative Permittivity, Constant. A dialog box appears.
3. Enter 1 for PERX (Relative permittivity), and click on OK. Material Model Number 1 appears in the Material Models Defined window on the left.
4. Choose menu path **Edit>Copy**. Click on OK to copy Material number 1 to Material number 2. Material Model Number 2 appears in the Material Models Defined window on the left.
5. In the Material Models Defined window, double-click on Material Model Number 2, and Permittivity (constant). A completed dialog box appears.
6. Replace the value in the PERX field with 10, and click on OK.
7. Choose menu path **Material>Exit** to remove the Define Material Model Behavior dialog box.
8. Click **SAVE_DB** on the ANSYS Toolbar.

[Back To Top](#)

Step 5: Create Model Geometry and Compress Numbers

1. Choose **Main Menu> Preprocessor> Modeling> Create> Areas> Rectangle> By Dimensions**. The Create Rectangle by Dimensions dialog box appears.
2. Enter the values shown below. (Use the TAB key to move between fields.)

| | |
|--------------|-------------|
| X1 field: 0 | Y1 field: 0 |
| X2 field: .5 | Y2 field: 0 |

3. Click Apply. The ANSYS Graphics Window displays the first rectangle.
4. To create a second rectangle, enter the following values:

| | |
|--------------|-------------|
| X1 field: .5 | Y1 field: 0 |
| X2 field: 5 | Y2 field: 1 |

5. Click Apply. The ANSYS Graphics Window displays the second rectangle.

6. To create a third rectangle, enter the following values:

X1 field: 0
X2 field: .5

Y1 field: 1
Y2 field: 10

7. Click Apply. The ANSYS Graphics Window displays the third rectangle.

8. To create a fourth rectangle, enter the following values:

X1 field: .5
X2 field: 5

Y1 field: 1
Y2 field: 10

9. Click OK. The ANSYS Graphics Window displays all four rectangles.

10. To glue all areas together, choose **Main Menu> Preprocessor> Modeling> Operate> Booleans> Glue> Areas**. A picking menu appears.

11. Click Pick All.

12. Choose **Main Menu> Preprocessor> Numbering Ctrls> Compress Numbers**. A dialog box appears.

13. Set the "Item to be compressed" field to "Areas."

14. Click OK.

[Back To Top](#)

Step 6: Apply Attributes to Model Regions and Prepare for Meshing

1. Choose **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
2. Change the top button from "Nodes" to "Areas."
3. Change the button just below it to "By Num/Pick."
4. Click OK. A picking menu appears.
5. Pick areas 1 and 2 by clicking on them. (Areas 1 and 2 are the two areas at the bottom of the Graphics Window.) The picked areas should change color.
6. Click OK.
7. Choose **Main Menu> Preprocessor> Meshing> Mesh Attributes> Picked Areas**. Click Pick All. The Area Attributes dialog box appears.
8. Set the "Material number" field to 2.
9. Click OK.

10. Choose **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
11. Check that the top two buttons are set to "Areas" and "By Num/Pick."
12. Click Sele All, then click OK. A picking menu appears.
13. Click Pick All.
14. Choose **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
15. Change the top button from "Areas" to "Lines."
16. Change the button below it to "By Location."
17. Click the Y Coordinates button on.
18. In the "Min, Max" field, enter 1.
19. Click Apply. ANSYS should respond by displaying a message about "2 lines" in the output window.
20. Click the Reselect and X Coordinates buttons on.
21. In the "Min, Max" field, enter .25.
22. Click OK. ANSYS should respond by displaying a message about "1 line" in the output window.

[**Back To Top**](#)

Step 7: Mesh the Model

1. Choose **Main Menu> Preprocessor> Meshing> Size Cntrs> Lines> All Lines**. The Element Sizes on All Selected Lines dialog box appears.
2. In the "No. of element divisions" field, enter 8.
3. Click OK.
4. Choose **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
5. Check that the top button is set to "Lines."
6. Change the second button to "By Num/Pick."
7. Click the From Full button on.
8. Click Sele All, then click OK. A picking menu appears.
9. Click Pick All.
10. Choose **Main Menu> Preprocessor> Meshing> MeshTool**. The MeshTool appears.

11. Click the Smart Size button on.
 12. Set the SmartSizing slider to 3.
 13. Click the Tri shape button on.
 14. Check that the Mesh button is set to "Areas."
 15. Click the MESH button. A picking menu appears.
 16. Click Pick All. The Graphics Window shows you the meshed model.
 17. Click Close to close the MeshTool.
-

[**Back To Top**](#)

Step 8: Apply Boundary Conditions and Loads

1. Choose **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
2. Change the top button to "Nodes."
3. Set the button just below it to "By Location."
4. Click the Y Coordinates and From Full buttons on.
5. In the "Min, Max" field, enter 1.
6. Click Apply.
7. Click the X Coordinates and Reselect buttons on.
8. In the "Min, Max" field, enter 0,.5.
9. Click OK.
10. Choose **Main Menu> Preprocessor> Loads> Define Loads> Apply> Electric> Boundary> Voltage> On Nodes**. A picking menu appears.
11. Click Pick All. The Apply VOLT on Nodes dialog box appears.
12. In the "Value of voltage (VOLT)" field, enter V1.
13. Click OK.
14. Choose **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
15. Check that the two top buttons are set to "Nodes" and "By Location."
16. Click the Y Coordinates and From Full buttons on.

17. In the "Min, Max" field, enter 0.
18. Click Apply.
19. Click the Also Sele button on.
20. In the "Min, Max" field, enter 10.
21. Click Apply.
22. Click the X Coordinates button on.
23. In the "Min, Max" field, enter 5.
24. Click OK.
25. Choose **Main Menu> Preprocessor> Loads> Define Loads> Apply> Electric> Boundary> Voltage> On Nodes**. A picking menu appears.
26. Click Pick All. The Apply VOLT on Nodes dialog box appears.
27. In the "Value of voltage (VOLT)" field, enter V0.
28. Click OK.

[**Back To Top**](#)

Step 9: Scale the Areas

1. Choose **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
2. Check that the top button is set to "Nodes," the next button is set to "By Num/Pick," and the "From Full" button is set on.
3. Click the Sele All button, then click OK. A picking menu appears. Click Pick All.
4. Choose **Main Menu> Preprocessor> Modeling> Operate> Scale> Areas**. A picking menu appears.
5. Click Pick All. The Scale Areas dialog box appears.
6. In the "RX, RY, RZ Scale Factors" fields, enter the following values:

RX field: .01

RY field: .01

RZ field: 0

7. In the "Items to be scaled" field, set the button to "Areas and mesh."
8. In the "Existing areas will be" field, set the button to "Moved."

9. Click OK.
 10. Choose **Main Menu> Finish.**
-

[**Back To Top**](#)

Step 10: Solve the Analysis

1. Choose **Main Menu> Solution> Solve> Current LS.** The Solve Current Load Step dialog box appears, along with a pop-up window listing the load step options.
 2. Review the pop-up window contents, then click Close.
 3. Click OK on the dialog box to start the solution. A pop-up message notifies you when solution is complete. Click Close.
 4. Choose **Main Menu> Finish.**
-

[**Back To Top**](#)

Step 11: Store Analysis Results

1. Choose **Main Menu> General Postproc> Element Table> Define Table.** The Element Table Data dialog box appears.
2. Click Add. The Define Additional Element Table Items dialog box appears.
3. In the "User label for item" field, enter SENE
4. In the scrollable lists in the "Results data item" field, highlight "Energy." (When you highlight "Energy" in the list on the left, "Elec energy SENE" will be highlighted in the list on the right automatically.)
5. Click OK.
6. Click Add.
7. In the "User label for item" field, enter EFX.
8. In the "Results data item" field, highlight "Flux & gradient" and "Elec field EFX." (You may have to scroll up to find the correct selections.)
9. Click OK.
10. Click Add.
11. In the "User label for item" field, enter EFY.
12. In the "Results data item" field, highlight "Flux & gradient" and "Elec field EFY."

13. Click OK. The Element Table Data dialog box now shows the SENE, EFX, and EFY items defined.
 14. Click Close to close the dialog box.
 15. Click SAVE_DB on the ANSYS Toolbar.
-

[Back To Top](#)

Step 12: Plot Analysis Results

1. Choose **Utility Menu> PlotCtrls> Numbering**. The Plot Numbering Controls dialog box appears.
 2. Set the "Numbering shown with" field to "Colors only."
 3. Click OK.
 4. Choose **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**. The Contour Nodal Solution Data dialog box appears.
 5. In the "Item to be contoured" field, highlight "DOF solution" and "Elec poten VOLT."
 6. Click OK. The Graphics Window displays a contour plot of equipotential lines.
 7. Choose **Main Menu> General Postproc> Plot Results> Vector Plot> User-defined**. The Vector Plot of User-defined Vectors dialog box appears.
 8. In the "Item" field, enter EFX.
 9. In the "Lab2" field, enter EFY
 10. Click OK. The Graphics Window displays a vector plot of the electric field.
-

[Back To Top](#)

Step 13: Perform Capacitance Calculations

1. Choose **Main Menu> General Postproc> Element Table> Sum of Each Item**. An informational dialog box appears.
2. Click OK. A pop-up window shows you all element table entries and their values.
3. Click Close to close the pop-up window.
4. Choose **Utility Menu> Parameters> Get Scalar Data**. The Get Scalar Data dialog box appears.
5. In the "Type of data to be retrieved" field, highlight "Results data" and "Elem table sums."
6. Click OK. The Get Element Table Sum Results dialog box appears.

7. In the "Name of parameter to be defined," field, enter W.
8. Set the "Element table item" field to "SENE."
9. Click OK.
10. Choose **Utility Menu> Parameters> Scalar Parameters**. The Scalar Parameters dialog box appears.
11. Type in the following values, pressing ENTER after typing each value:
 $C = (w^2)/((V1-V0)^2)$
 $C = ((C^2)*1e12)$
12. Click Close.
13. Choose **Utility Menu> List> Status> Parameters> Named Parameter**. The Named Parameter Status dialog box appears.
14. In the "Name of parameter" field, highlight C.
15. Click OK. A pop-up window displays the value of the C parameter (capacitance).
16. Click Close to close the pop-up window.

[**Back To Top**](#)

Step 14: Finish the Analysis

To finish the analysis, choose **Main Menu> Finish**. Then click QUIT on the ANSYS Toolbar. Choose an exit option and click OK.

[Low-Frequency Guide](#) | [Chapter 13. Electrostatic Field Analysis \(h-Method\)](#) |

13.6. Doing an Electrostatic Analysis (Command Method)

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

You can perform the example analysis of the microstrip transmission line using the ANSYS commands shown below instead of GUI menu choices. All text prefaced by an exclamation point (!) is a comment.

```

/BATCH,LIST
/PREP7
/TITLE, MICROSTRIP TRANSMISSION LINE ANALYSIS
ET,1,PLANE121                      ! USE 2-D 8-NODE ELECTROSTATIC ELEMENT
V1=1.5                                ! DEFINE STRIP POTENTIAL
V0=0.5                                ! DEFINE GROUND POTENTIAL
MP,PERX,1,1                            ! FREE SPACE RELATIVE PERMITTIVITY
MP,PERX,2,10                           ! SUBSTRATE RELATIVE PERMITTIVITY
RECTNG,0,.5,0,1
RECTNG,.5,5,0,1
RECTNG,0,.5,1,10
RECTNG,.5,5,1,10
AGLUE,ALL
NUMCMP,AREA
ASEL,S,AREA,,1,2

AATT,2
ASEL,ALL                               ! SET AREA ATTRIBUTES FOR AIR
LSEL,S,LOC,Y,1
LSEL,R,LOC,X,.25
LESIZE,ALL,,,8
LSEL,ALL
SMRTSIZE,3
MSHAPE,1                                ! Triangle mesh
AMESH,ALL
NSEL,S,LOC,Y,1                          ! SELECT NODES ON MICROSTRIP
NSEL,R,LOC,X,0,.5
D,ALL,VOLT,V1
NSEL,S,LOC,Y,0
NSEL,A,LOC,Y,10
NSEL,A,LOC,X,5                          ! SELECT EXTERIOR NODES
D,ALL,VOLT,V0                          ! APPLY GROUND POTENTIAL
NSEL,ALL
ARSCALE,ALL,,,01,.01,0,,0,1            ! SCALE MODEL TO METERS
FINISH
/SOLUTION
SOLVE
FINISH
/POST1
ETABLE,SENE,SENE                        ! STORE ELECTROSTATIC ENERGY
ETABLE,EFX,EF,X                         ! STORE POTENTIAL FIELD GRADIENTS
ETABLE,EFY,EF,Y
/NUMBER,1
PLNSOL,VOLT
PLVECT,EFX,EFY                          ! DISPLAY EQUIPOTENTIAL LINES
                                         ! DISPLAY VECTOR ELECTRIC FIELD (VECTOR)

```

```
SSUM                                ! SUM ENERGY
*GET,W,SSUM,,ITEM,SENE             ! GET ENERGY AS W
C=(W*2)/((V1-V0)**2)               ! CALCULATE CAPACITANCE (F/M)

C=((C*2)*1E12)                     ! FULL GEOMETRY CAPACITANCE (PF/M)
*STATUS,C                           ! DISPLAY CAPACITANCE
FINISH
```

[Low-Frequency Guide](#) | [Chapter 13. Electrostatic Field Analysis \(h-Method\)](#) |

13.7. Doing an Example Capacitance Calculation (Command Method)

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

The following is an example of how to perform a capacitance matrix calculation by issuing ANSYS commands. You can also perform the analysis through the ANSYS GUI menus.

For details on extracting capacitance from multi-conductor systems, see [Extracting Capacitance from Multi-conductor Systems](#) in this manual.

13.7.1. The Example Described

In this example, two long cylinders sit above an infinite ground plane. The objective is to compute the self and mutual capacitance coefficients between the conductors and ground.

13.7.2. Modeling Notes

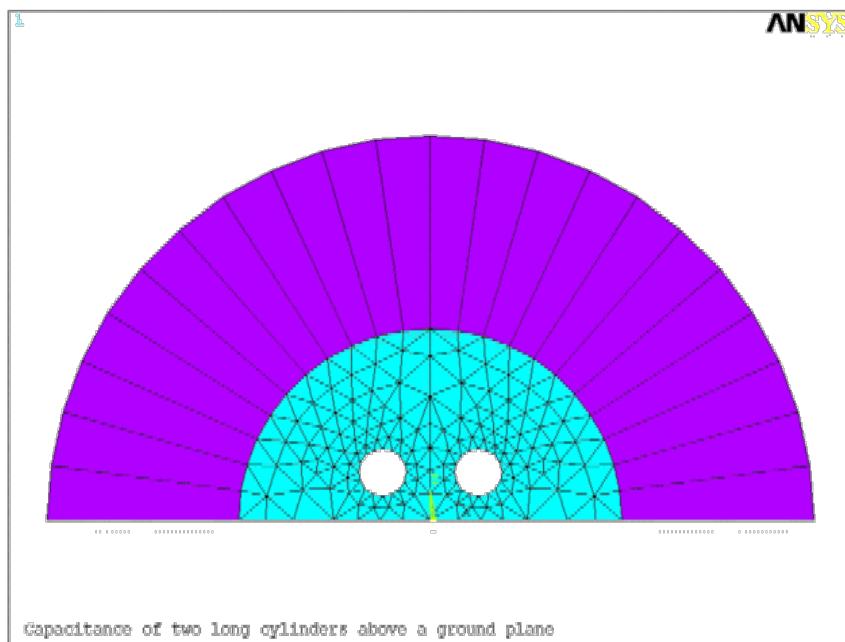
The ground plane and the Infinite elements share a common boundary at the outer radius of the model. The Infinite elements by nature can only represent a zero potential at the infinite location. Since the ground plane shares a common boundary with the infinite elements, they together represent the ground conductor. The nodes of the ground plane are sufficient to represent the ground conductor since the infinite element nodes are internally grounded by the program. By grouping the other two cylindrical conductor node sets into node components as well, you arrive at a 3-conductor system.

[Figure 13.6: "Model Areas of Capacitance Example Problem"](#) displays the model areas and [Figure 13.7: "Elements of Capacitance Example Problem"](#) displays the finite elements.

Figure 13.6 Model Areas of Capacitance Example Problem



Figure 13.7 Elements of Capacitance Example Problem



Capacitance of two long cylinders above a ground plane

13.7.3. Computed Results

The computed ground and lumped capacitance results for this example problem are as follows:

$$\begin{aligned} (C_g)_{11} &= 0.454E-4 \text{ pF} & (C_l)_{11} &= 0.354E-4 \text{ pF} \\ (C_g)_{12} &= -0.998E-5 \text{ pF} & (C_l)_{12} &= 0.998E-5 \text{ pF} \\ (C_g)_{22} &= 0.454E-4 \text{ pF} & (C_l)_{22} &= 0.354E-4 \text{ pF} \end{aligned}$$

13.7.4. Command Listing

You can perform this example capacitance matrix calculation using the ANSYS commands shown below. Text prefaced by an exclamation point (!) is a comment.

```

/batch,list
/prep7
/title, Capacitance of two long cylinders above a ground plane

a=100           ! Cylinder inside radius (μm)
d=400           ! Outer radius of air region
ro=800          ! Outer radius of infinite elements

et,1,121        ! 8-node 2-D electrostatic element
et,2,110,1,1    ! 8-node 2-D Infinite element
emunit,epzro,8.854e-6 ! Set free-space permittivity for μMKS units
mp,perx,1,1

cyl4,d/2,d/2,a,0      ! Create mode in first quadrant
cyl4,0,0,ro,0,,90
cyl4,0,0,2*ro,0,,90

aovlap,all
numcmp,area
smrtsiz,4
mshape,1          ! Mesh air region
amesh,3

lsel,s,loc,x,1.5*ro
lsel,a,loc,y,1.5*ro
lesize,all,,,1
type,2
mshape,0
mshkey,1
amesh,2          ! Mesh infinite region
arsym,x,all      ! Reflect model about y axis
nummrg,node
nummrg,kpoi

csys,1

nSEL,s,loc,x,2*ro
sf,all,inf        ! Set infinite flag in Infinite elements

```

```
local,11,1,d/2,d/2
nsel,s,loc,x,a
cm,cond1,node
local,12,1,-d/2,d/2
nsel,s,loc,x,a
cm,cond2,node
csys,0
nsel,s,loc,y,0
cm,cond3,node
allsel,all
finish
/solu
cmatrix,1,'cond',3,0      ! Compute capacitance matrix coefficients
finish
```

[Low-Frequency Guide](#) | [Chapter 13. Electrostatic Field Analysis \(h-Method\)](#) |

13.8. Doing an Electrostatic Analysis Using Trefftz Method (Command Method)

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

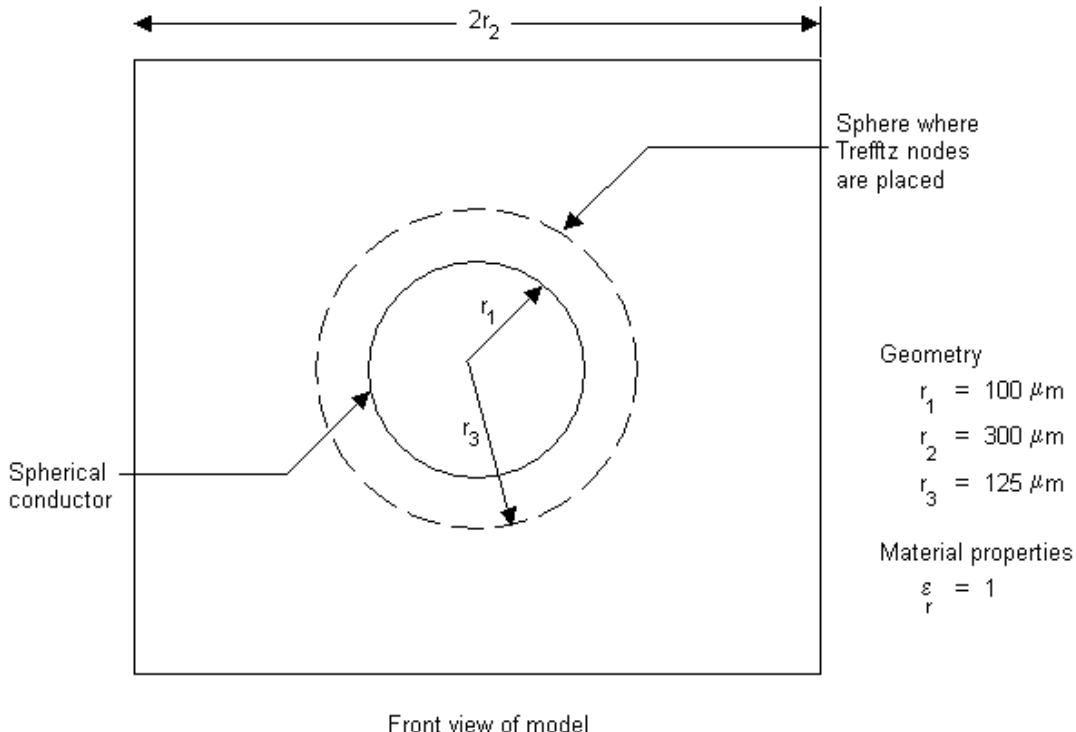
The following is an example of how to create a Trefftz domain to solve an open boundary problem in electrostatics. You can also perform the analysis through the ANSYS GUI menus.

For details on the Trefftz methodology, see [Trefftz Method for Open Boundary Representation](#) of this manual.

13.8.1. The Example Described

This is an analysis of a charged spherical conductor in free-space as shown in [Figure 13.8: "Charged Spherical Conductor"](#). The objective is to compute the capacitance.

Figure 13.8 Charged Spherical Conductor



13.8.2. Modeling Notes

For this problem, you build a model by subtracting a spherical volume from a solid cube (representing the modeled finite element air region). The nodes at the exterior of the sphere represent the conductor nodes. You then create Trefftz source nodes ([TZAMESH](#)) by defining another spherical volume placed between the spherical boundary of the conductor and the exterior volume of air. After creating the Trefftz source nodes, you generate a Trefftz domain using the [TZEGEN](#) command. You then compute the capacitance using the [CMATRIX](#) command macro.

13.8.3. Expected Results

The target capacitance is 0.0111 pF.

13.8.4. Command Listing

You can perform this example analysis of a charged spherical conductor in free-space using the ANSYS commands shown below. Text prefaced by an exclamation point (!) is a comment.

```

/batch,list
/title, Sphere to infinity capacitance using a Trefftz Domain
/com =====
/com
/com Analytical solution for capacitance is
/com
/com      Cself = 4*Pi*Eps0*EpsR*R1
/com
/com =====
!
N=5                      ! subdivision parameter
R1=100                   ! radius of the sphere (microns)
R2=300                   ! half side of the cube
R3=125                   ! radius for Trefftz nodes
!
/PREP7                   ! Enter Preprocessor
!
et,1,123                ! 10 node tetrahedral
!
emunit,epzro,8.854e-6    ! free space permittivity (µMKS units)
mp,perx,1,1               ! relative permittivity
!
sphere,0,R1,0,360        ! conductor outer radius
/view,1,1,1,1
/replot
block,-R2,R2,-R2,R2,-R2,+R2   ! exterior FE air region (cube)
vsbv,2,1
!
nummrg,all
!
mshape,1                 ! mesh with tets
mshkey,0                 ! free meshing
esize,,n                  ! mesh control
!
VMESH,ALL
cm,vol,volu
sphere,125
cmsel,u,vol
cm,tvol,volu
!
nsel,s,loc,x,R2          ! select outer nodes of FE domain
nsel,a,loc,x,-R2
nsel,a,loc,y,R2
nsel,a,loc,y,-R2
nsel,a,loc,z,R2
nsel,a,loc,z,-R2
sf,all,inf                ! infinite surface flag on FE exterior
!
csys,2
nsel,s,loc,x,r1
cm,cond1,node
csys,0
allsel
tzamesh,'tvol',,2         ! Create Trefftz nodes
tzegen                     ! Create Trefftz Domain
!
```

```
finish
!
/solu
antyp,static
eqslv,jcg           ! select JCG solver
!
! Compute capacitance
!
cmatrix,1,'cond',1,1      ! symmetry=1, no cond=1, ground at infinity
finish
```

[Low-Frequency Guide](#) | [Chapter 13. Electrostatic Field Analysis \(h-Method\)](#) |

13.9. Doing an Electrostatic Analysis Using Trefftz Method (GUI Method)

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

As shown below, you can use GUI menu paths to perform the example analysis of creating a Trefftz domain to solve an open boundary problem in electrostatics.

Step 1: Start the Analysis

1. Activate the ANSYS launcher.
2. Select the ANSYS simulation environment, choose your license, and click **Run**.
3. When the Graphical User Interface is fully active, choose **Utility Menu> File> Change Title**. A dialog box appears.
4. Enter the title text, Sphere to Infinity Capacitance
5. Click on OK.
6. Choose **Main Menu> Preferences**. The Preference dialog box appears.
7. Click **Electric** on.
8. Click on OK.

[Back To Top](#)

Step 2: Define Analysis Parameters

1. Choose menu path **Utility Menu> Parameters> Scalar Parameters**. The Parameters dialog box appears.
2. Type in the parameter values shown below. (Press ENTER after entering each value.)

n = 5
r1 = 100
r2 = 300
r3 = 125

3. Click on Close to close the dialog box.

Step 3: Define Element Types

1. Choose **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**. The Element Types dialog box appears.
 2. Click on Add. The Library of Element Types dialog box appears.
 3. In the scrollable fields, click on (highlight) Electrostatic and 3D Tet 123 ([SOLID123](#)).
 4. Click on OK. ANSYS returns you to the Element Types dialog box.
 5. Click on Close to close the Element Types dialog box.
-

Step 4: Define Material Properties

1. Choose **Main Menu> Preprocessor> Material Props> Electromag Units**. The Electromagnetic Units dialog box appears.
 2. Select the User-Defined option button and click on OK. A Electromagnetics Units text entry box appears.
 3. Enter 8.854e-6 for the free-space permittivity ($\mu\text{MKS}V$ units) and click on OK.
 4. Choose **Main Menu> Preprocessor> Material Props> Material Models**. The Define Material Model Behavior dialog box appears.
 5. In the Material Models Available window, double-click on the following options: Electromagnetics, Relative Permittivity, Constant. A dialog box appears.
 6. Enter 1 for PERX (Relative permittivity), and click on OK. Material Model Number 1 appears in the Material Models Defined window on the left.
 7. Choose menu path **Material>Exit** to remove the Define Material Model Behavior dialog box.
-

Step 5: Create Solid Model

1. Choose **Main Menu> Preprocessor> Modeling> Create> Volumes> Sphere> By Dimensions**. The Create Sphere by Dimensions dialog box appears.
2. In the “RAD1” field, enter r1, in the “RAD2” field, enter 0, in the “THETA1” field, enter 0, in the “THETA2” field, enter 90, and click on OK.
3. Choose **Utility Menu> PlotCtrls> Pan, Zoom, Rotate**. A Pan-Zoom-Rotate dialog box appears.

4. Click on the Iso button, then click on Close. The ANSYS Graphics Window displays the block.
5. Choose **Main Menu> Preprocessor> Modeling> Create> Volumes> Block> By Dimensions**. The create Block by Dimensions dialog box appears.
6. In the “X coordinates” field, enter 0, r2, in the “Y coordinates” field, enter 0, r2, and in the “Z coordinates” field, enter 0, r2.
7. Choose **Main Menu> Preprocessor> Modeling> Operate> Booleans> Subtract> Volumes**. The Subtract Volumes picking menu appears.
8. Pick (or enter in the picker) the block volume (volume 2) and click on OK. The Subtract Volumes picking menu reappears.
9. Pick (or enter in the picker) the spherical volume (volume 1) and click on OK.
10. Choose **Main Menu> Preprocessor> Modeling> Reflect> Volumes**. The Reflected Volumes picking menu appears.
11. Click on Pick All. The Reflected Volumes dialog box appears.
12. Check that the Y-Z plane is selected and click on OK.
13. Choose **Main Menu> Preprocessor> Modeling> Reflect> Volumes**. The Reflected Volumes picking menu appears.
14. Click on Pick All. The Reflected Volumes dialog box appears.
15. Select the X-Z plane and click on OK.
16. Choose **Main Menu> Preprocessor> Modeling> Reflect> Volumes**. The Reflected Volumes picking menu appears.
17. Click on Pick All. The Reflected Volumes dialog box appears.
18. Select the X-Y plane and click on OK.
19. Choose **Main Menu> Preprocessor> Numbering Ctrs> Merge Items**. A Merge Coincident or Equivalently Defined Items dialog box appears.
20. Set the “Type of Item to be merge (Label)” to All and click on OK.

[Back To Top](#)

Step 6: Mesh the Model and Create a Finite Element Model Component

1. Choose **Main Menu> Preprocessor> Meshing> MeshTool**. The MeshTool appears.
2. Click on the Set Global Size Controls button. A Global Element Sizes dialog box appears.

3. In the “No. of Element Divisions (NDIV)” field, enter n, and click on OK.
 4. Make sure that Volumes, Tet, and Free are selected in the MeshTool.
 5. Click MESH. The Mesh Volumes picking menu appears.
 6. Click on Pick All. The display changes to show the meshed volumes. Click Close to close the MeshTool.
 7. Choose menu path **Utility Menu> Select> Comp/Assembly> Create Component**. The Create Component dialog box appears.
 8. In the “Component Name (Cname)” field enter vol, set the “Component is made of (Entity)” to volumes, and click on OK.
-

[**Back To Top**](#)

Step 7: Apply Infinite Surface Flag to Exterior Surface of Finite Element Region

1. Choose menu path **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
 2. Set the top box to nodes and the second box to By Location. Enter r2 in the “Min/Max” field, select X coordinates, and click on OK.
 3. Choose menu path **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
 4. Enter -r2 in the “Min/Max” field, change the selection type radio button to Also Sele, and click on Apply.
 5. Enter r2 in the “Min/Max” field, change the coordinates radio button to Y coordinates, and click on Apply.
 6. Enter -r2 in the “Min/Max” and click on Apply.
 7. Enter r2 in the “Min/Max” field, change the coordinates radio button to Z coordinates, and click on Apply.
 8. Enter -r2 in the “Min/Max” field and click on OK.
 9. Choose menu path **Main Menu> Preprocessor> Loads> Define Loads> Apply> Electric> Flag> Infinite Surf> On Nodes**. The Apply INF on Nodes dialog box appears.
 10. Click on Pick All.
-

[**Back To Top**](#)

Step 8: Create a Spherical Conductor Surface Node Component

1. Choose menu path **Utility Menu> WorkPlane> Change Active CS to> Global Spherical**.
2. Choose menu path **Utility Menu> Select> Entities**. The Select Entities dialog box appears.

3. Check that the top box is set to nodes and the second box is set to By Location. Change the coordinates radio button to X coordinates, change the selection type radio button to From Full, enter r1 in the “Min/Max” field, and click on OK.
 4. Choose menu path **Utility Menu> Select> Comp/Assembly> Create Component**. The Create Component dialog box appears.
 5. In the “Component Name (Cname)” field enter cond1, set the “Component is made of (Entity)” to nodes, and click on OK.
-

[**Back To Top**](#)

Step 9: Create Trefftz Nodes and Trefftz Domain

1. Choose menu path **Utility Menu> WorkPlane > Change Active CS to> Global Cartesian**.
 2. Choose menu path **Main Menu> Preprocessor> Trefftz Domain> TZ Geometry> Create> Volume> Sphere> By Dimensions**. The Create Sphere by Dimensions dialog box appears.
 3. In the “Outer Radius (RAD1)” field enter r3. Check that the “Starting angle (THETA1) and Ending angle (THETA2) are 0 and 360 degrees, respectively. Click on OK.
 4. Choose menu path **Utility Menu> Select> Comp/Assembly> Select Comp/Assembly**. The Select Component or Assembly dialog box appears.
 5. In the “Select Entities belonging to Component or Assembly (CMSEL)” field choose VOL, in the “Type of Selection (Type)” field choose Unselect, and click on OK.
 6. Choose menu path **Main Menu> Preprocessor> Trefftz Domain> Mesh TZ Geometry**. The Mesh TZ Geometry dialog box appears.
 7. Pick the Trefftz volume (volume 5) and click on OK. The Parameters for mesh of Trefftz domain dialog box appears.
 8. Check that the No. of element divisions (NDIV) is set to 2 and click on OK. A pop-up window displays the results for Trefftz nodes.
 9. Review the results, which show 20 Trefftz nodes have been created, and click on Close.
 10. Choose menu path **Utility Menu> Select> Everything**.
 11. **Main Menu> Preprocessor> Trefftz Domain> Superelement> Generate TZ** A pop-up window displays the results for the Trefftz Domain.
 12. Review the results and click on Close.
-

[**Back To Top**](#)

Step 10: Compute Capacitance

1. Choose menu path **Main Menu> Finish**.
 2. Choose menu path **Main Menu> Solution> Analysis Type> New Analysis**. A New Analysis dialog box appears with Steady-State entered for the “Type of Analysis (ANTYPE).”
 3. Click on OK.
 4. Choose menu path **Main Menu> Solution> Analysis Type> Analysis Options**. The Static or Steady-State Analysis dialog box appears.
 5. Set the “Equation Solver (EQSLV)” to Jacobi Conj Grad and click on OK.
 6. Choose menu path **Main Menu> Solution> Solve> Electromagnet> Static Analysis> Capac Matrix**. The Capac Matrix dialog box appears.
 7. In the “Symfac Geometric symmetry factor (Symfac)” field, enter 1, in the “Component name identifier (Condname)” field, enter cond, in the “Number of cond components (Ncond)” field, enter 1, in the “Ground Key (Grndkey)” field, enter 1, and click on OK.
 8. Review the results and click on Close.
 9. Choose menu path **Main Menu> Finish**.
-
-

[Low-Frequency Guide](#) |

Chapter 14. p-Method Electrostatic Analysis

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

The p-method obtains results such as potential (voltage), electric field, electric flux density, electrostatic force or energy to your required degree of accuracy. To calculate these results, the p-method employs higher order polynomial levels (p-levels) of the finite element shape functions to approximate the real solution.

This feature works by taking a finite element mesh, solving it at a given p-level, increasing the p-level selectively, and then solving the mesh again. After each iteration the results are compared for convergence against a set of convergence criteria. You can specify the convergence criteria to include potential, electric field, or electric flux density at a point (or points) in the model, global-stored energy and global forces on a body (Maxwell Stress Tensor). The higher the p-level, the better the finite element approximation to the real solution.

In order to capitalize on the p-method functionality, you don't have to work only within the confines of p-generated meshes. The p-method is most efficient when meshes are generated considering that p-elements will be used, but this is not a requirement. Of course, you might want to create and mesh your model using p-elements, but you can also perform a p-method solution using meshes that have been generated with h-elements (generated by ANSYS or your CAD package), if the elements are at least mid-noded. This provides you with the flexibility of taking advantage of the p-method solution option independently of how the mesh was created. The p-method can improve the results for any mesh automatically.

The following p-Method electrostatic analysis topics are available:

- [Benefits of Using the p-Method](#)
 - [Using the p-Method](#)
 - [Doing an Example p-Electrostatic Analysis \(Command Method\)](#)
-
-

[Low-Frequency Guide](#) | [Chapter 14. p-Method Electrostatic Analysis](#) |

14.1. Benefits of Using the p-Method

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

The p-method solution option offers many benefits for electrostatic analyses that are not available with the traditional h-method, discussed in "[Electrostatic Field Analysis \(h-Method\)](#)". The most convenient benefit is the ability to obtain good results to a desired level of accuracy without rigorous user-defined meshing controls. If you are new to finite element analysis or do not have a solid background in mesh design, you might prefer this method since it relieves you of the task of manually designing an accurate mesh.

In addition, the p-method adaptive refinement procedure offers error estimates that are more precise than those of the h-method, and can be calculated locally as well as globally (for example, total force on a body rather than electrostatic energy). For example, if you need to obtain highly accurate solutions at a point, such as for dielectric breakdown, or forces on a body, the p-method offers an excellent means of obtaining these results to the required accuracy.

[Low-Frequency Guide](#) | [Chapter 14. p-Method Electrostatic Analysis](#) |

14.2. Using the p-Method

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

The procedure for a p-method static analysis consists of four main steps:

1. Select the p-method procedure.
2. Build the model.
3. Apply loads and obtain the solution.
4. Review the results.

Each step is discussed in detail in the following sections.

14.2.1. Select the p-Method Procedure

You can activate the p-method solution procedure in two ways: through the GUI or by defining a p-element [[ET](#)].

- *Activating p-method through the GUI:*

Command(s): [/PMETH](#)

GUI: Main Menu> Preferences> p-method Electr.

- *Defining a p-element:* The p-method solution procedure can also be activated by defining a p-element. If you are working outside of the GUI, the definition of a p-element lets the program know that a p-method solution is to be done; no other commands are necessary to initiate p-method. From within the GUI, you can also issue the [ET](#) command in the "Input Window" to activate the p-method procedure. (Remember, the [ET](#) command must be entered in the "Input Window," since, by default, only h-elements are displayed in the GUI unless p-method is active.)

Command(s): [ET](#)

GUI: Main Menu> Preprocessor> Element Type> Add/Edit/Delete

14.2.2. Build the Model

To build a model with p-elements, you may follow the procedure listed below.

1. Define the element types.

2. Specify material properties.
3. Define the model geometry.
4. Mesh the model into elements.

The above steps are common to most analyses. The [Modeling and Meshing Guide](#) explains those steps in detail. In this section we will explain the techniques that are unique to a p-analysis.

14.2.2.1. Define the Element Types

You can use the following two p-elements to build your model:

Table 14.1 Element Types

| Element | Dimens. | Shape or Characteristic | DOFs |
|--------------------------|---------|-------------------------|----------------------|
| SOLID127 | 3-D | Tetrahedral, 10 nodes | Voltage at each node |
| SOLID128 | 3-D | Hexahedral, 20 nodes | Voltage at each node |

Note

h-elements and p-elements cannot be active at the same time in your model (except for the [MATRIX50](#) element use as a superelement for a Trefftz domain).

14.2.2.1.1. Specifying a p-Level Range

Various options are available for use with p-elements. One important option is the ability to specify, either locally or globally, a range in which the p-level may vary.

The range within which the p-level may vary can be controlled locally through the p-element KEYOPT settings (KEYOPT(1) and KEYOPT(2)), or globally across the entire model with [PPRANGE](#). By default, the p-level range is 2 to 8.

When both KEYOPT values and [PPRANGE](#) have been used to specify p-level ranges, the local p-level range set by KEYOPT(1) and KEYOPT(2) will take precedence over the global p-level range [[PPRANGE](#)].

For example, if you set a global p-level range between 3 and 8 with [PPRANGE](#), then define a local p-level range of 4 to 6 for [SOLID127](#) elements ([ET,1,127,4,6](#)), the p-level for the [SOLID127](#) elements may only vary between 4 and 6, while the rest of the model may vary between 3 and 8.

At the (default) starting p-level of 2, convergence checking is performed ([PEMOPTS](#) command) to determine those elements which are converged and may have their p-level fixed at 2. That is, these elements will remain at a p-level of 2, and will be eliminated from any further convergence checking. Additional checking is performed at each iteration to fix the p-levels of the elements which are converged.

Use local p-range control to eliminate regions of little importance from high p-escalation. Use global p-range control for overall control of the p-level. These range controls are not necessary, but p-escalations to high p-levels increase CPU run time. Therefore, it is advantageous to have such controls available.

- Defining a local p-level range:

Command(s): [ET](#),ITYPE,Ename,KOP1,KOP2

GUI: Main Menu> Preprocessor> Element Type> Add/Edit/Delete

- Defining a global p-level range:

Command(s): [PPRANGE](#)

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> p-Method> Set p range

See the [Elements Reference](#) for complete descriptions of each of the above element types.

14.2.2.2. Specify Material Properties and/or Real Constants

14.2.2.2.1. Units

You may solve electrostatic field problems in a variety of units. ANSYS requires that all geometry dimensions, properties, and input loads (excitations) be consistent with respect to a units system. By default, ANSYS supports the MKS (or MKSV) system of units (meter, kilogram, second, volt, ampere). For microsystems, it may be more advantageous to work in other systems of units such as a μ MKS (micrometer, kilogram, second, volt, pico-ampere) or a μ MSVfA (micrometer, second, volt, femto-ampere, gram). For electrostatic analysis, you must select an appropriate value of free-space permittivity consistent with the system of units to be used. This is done via the [EMUNIT](#) command. By default, the free-space permittivity is 8.854e-12 Farads/meter (MKS units). For μ MKS units, you should use 8.854e-6 pico-Farads/micro-meter. For μ MSVfA units, you should use 8.854e-3 femto-Farads/micro-meter. See [Building the Model](#) for a more complete description of alternate systems of units.

To specify a system of units, issue one of the following:

Command(s): [EMUNIT](#)

GUI: Main Menu> Preprocessor> Material Props> Electromag Units

14.2.2.2.2. Material Properties

Material properties for p-elements (relative permittivity) may be either constant or temperature-dependent, as well as isotropic or orthotropic.

To define relative permittivity, issue one of the following:

Command(s): [MP](#)

GUI: Main Menu> Preprocessor> Material Props> Material Models> Electromagnetics> Relative Permittivity> Isotropic

Element coordinate systems [[ESYS](#)] may be used for orthotropic material directions. All element results in POST1 however may only be viewed in Global Cartesian Coordinates.

14.2.2.3. Define the Model Geometry

You can create your model using any of the various techniques outlined in the [Modeling and Meshing Guide](#), or you can import it from a CAD system. If you are generating your model from within ANSYS, you can use either solid modeling or direct generation techniques.

Note

Using direct generation is not recommended when you plan to create a p-mesh since all p-elements require that midside nodes be included in their geometric definition. In cases where surface curvature is important, it would not only be tedious, but possibly imprecise to manually define each midside node. In addition, the **EMID** command does not place nodes on a curved line. It is much more convenient to let the program generate the midside nodes using solid modeling.

You may not drop any midside nodes from p-elements.

[Low-Frequency Guide](#) | [Chapter 14. p-Method Electrostatic Analysis](#) |

14.3. Doing an Example p-Electrostatic Analysis (Command Method)

[www.kxcad.net Home](#) > [CAE Index](#) > [ANSYS Index](#) > [Release 11.0 Documentation for ANSYS](#)

This section demonstrates how to do a simple electrostatic analysis using the p-method described in the previous sections.

14.3.1. The Example Described

This is an analysis of a 2-D electrostatic comb drive with one set of fingers. The moving finger (beam) has an applied voltage of 10, while the U-shaped finger is fixed and grounded. The objective is to calculate the electrostatic driving force.

The dimensions are:

$$tt = 40 \mu\text{m}$$

$$wf = 2\mu\text{m}$$

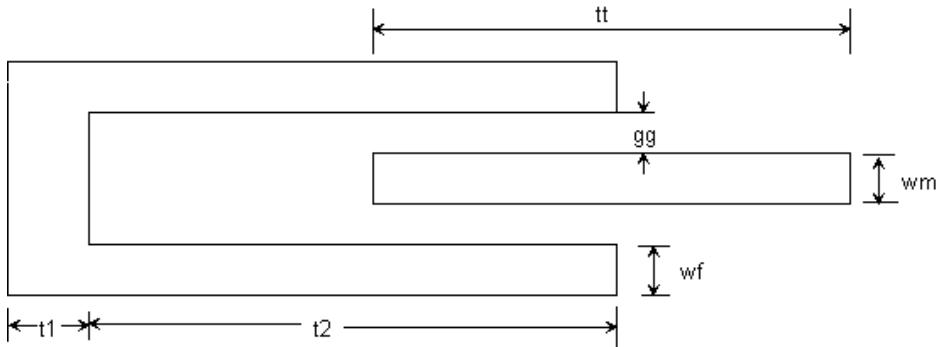
$$gg = 2 \mu\text{m}$$

$$t1 = 4 \mu\text{m}$$

$$wm = 2 \mu\text{m}$$

$$t2 = 40 \mu\text{m}$$

Figure 14.9 Electrostatic Comb Drive



14.3.2. Modeling Notes and Results

The mesh is shown in [Figure 14.10: "Wedge Element Mesh"](#).

Figure 14.10 Wedge Element Mesh



The p-method requires that the error for the electrostatic force acting on the moving finger be within 1% between consecutive p-loops. [Figure 14.11: "p-Convergence Performance for Electrostatic Force"](#) shows convergence results. The force predicted by the p-method is 27% higher than that obtained with the initial mesh.

Figure 14.11 p-Convergence Performance for Electrostatic Force



14.3.3. Command Listing

You can perform this example analysis of a 2-D electrostatic comb drive with one set of fingers using the ANSYS commands shown below. Text prefaced by an exclamation point (!) is a comment.

```

/batch,list
/show,combp,grph]
/title, Comb Drive Example
/prep7
! dimensions (microns) / parameters
wf=2                      ! Fixed finger width
wm=2                      ! Moving finger width
gg=2                      ! Gap
tt=40                     ! Moving length
t1=4                      ! Fixed finger backing
t2=tt                     ! Moving finger length
hh=10                     ! Height
dt=tt/2
V0=10
xx=(2*tt-dt)*2.75
y1=(wm/2)+gg
y2=(wm/2)+gg+wf
x1=tt+t1
! Create Model
cyl4,0,0,xx,0,,360,1
block,-tt,0,y1,y2,0,1
block,(-tt-t1),-tt,-y2,y2,0,1
block,-tt,0,-y1,-y2,0,1
block,-dt,(tt-dt),(-wm/2),(wm/2),0,1
block,(-dt-(gg/2)),(tt-dt+(gg/2)),((-wm/2)-(gg/2)),(wm/2)+(gg/2),0,1
dx=wm*1.1
dy=dx
block,-x1-dx,tt-ddt+dx,-y2-dy,y2+dy,0,1
vooverlap,all
/com -----
/com Volume Component
/com -----
/com 2,8,9    Fixed Finger
/com 5        Moving Finger
/com 11       Air Layer around Moving Finger
/com 12       Air Layer around the Comb Drive
/com 10       Boundary
/com -----
! 2D mesh
et,11,200,7                 ! Use high order tri mesh element
esize,gg/2
mshap,1                      ! Force a triangle mesh
amesh,52                      ! Mesh air layer around moving finger
esize,gg
amesh,54                      ! Mesh air layer around comb drive
esize,(y2+dy)*5
amesh,50                      ! Mesh boundary
! 3D mesh
et,1,128
emunit,epzro,8.854e-6         ! Free space permittivity (uMKSV units)
mp,perx,1,1                   ! Set relative permittivity to 1
mat,1
lsel,s,loc,z,.5
lesize,all,,,1                 ! Set element size in z-direction to 1
lsel,all
type,1
vsel,s,volu,,10,12,1
vsweep,all                     ! Create wedge volume elements from tri mesh
vsel,all

```

```

vsel,s,volu,,10          ! Plot model
eslv,s
eplot
allsel
asel,s,loc,z,1           ! Select original areas
aclear,all
etdelete,11                ! Remove meshing elements
asel,all
csys,1
nsel,s,loc,x,xx
d,all,VOLT,0              ! Apply volt=0 at outer boundary
nsel,all
csys,0
vsel,s,volu,,5
cm,MFINGER,volu
asel,s,ext
nsla,s,1
d,all,VOLT,0              ! Apply a zero potential on the moving finger
sf,all,mxwf
vsel,s,volu,,2
vsel,a,volu,,8,9,1
cm,FFINGER,volu
asel,s,ext
nsla,s,1
d,all,VOLT,V0             ! Apply 10 Volts on the fixed finger
allsel
! p-convergence control
pemopt,5.0,dual
pconv,2,efor,x
/gst,on
save
finish
/solu
eqslv,iccg                 ! Set ICCG solver
solve
finish
/post1
set,last                     ! Set the last data step
vsel,s,volu,,11,12
eslv,s
pplot                         ! Plot the p-levels around the fingers
esel,all
plconv,all
etable,fx,fmag,x
etable,fy,fmag,y
etable,fz,fmag,z
ssum
*get,fx,ssum,,item,fx        ! Get summed forces
*get,fy,ssum,,item,fy
*get,fz,ssum,,item,fz
/com,Comb Drive Force
*stat,fx
finish

```
