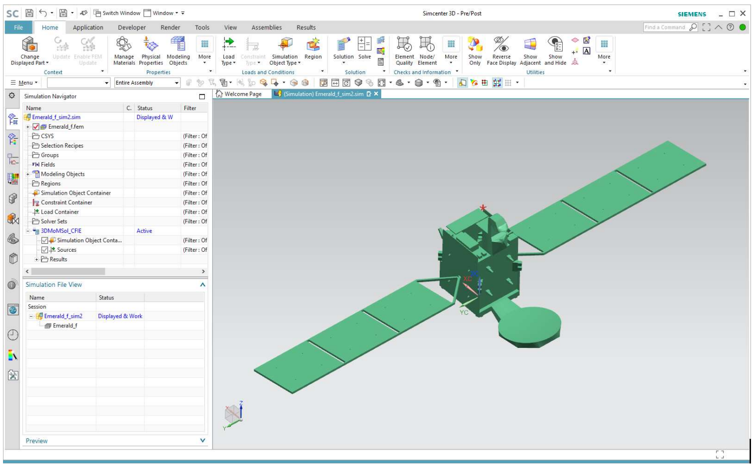
Rev. 1.1

N°doc:MN/2019/014

Config.: SimcenterEM-PRCS-OUT-MN

Simcenter 3D High Frequency EM



Simcenter 3D High Frequency EM v.1.0 – User Manual

Pisa, September 2019

Tot pag. N°. = 195

KEYWORDS SOFTWARE, FRAMEWORK, CODE, ELECTROMAGNETIC, 3D MOM, 2.5D MOM, S-PEEC, UTD, IPO, ANTENNA MODELLING, ANTENNA SITING

SUMMARY The purpose of this document is to describe the distinctive features, modelling capabilities and working procedures of *Simcenter 3D HFEM*.

Document Evolution		
Revision	Date	Reason of change
Rev. 1.0	April 2019	First Edition
Rev. 1.1	September 2019	Revision

Document Change Record (Log)		
RNC	*Reference*	*Description of changes*

SW Versions covered by this document
Simcenter3D_HFEM v.01.00

Disclaimer
IDS will not be responsible for the consequences caused by the improper use of the software.

Contacts
IDS Ingegneria dei Sistemi S.p.A. Via Enrica Calabresi, 24 Loc. Montacchiello 56121 PISA - ITALY Tel: +39-050-3124-1 Fax: +39-050-3124-201

CONTENTS

FIGURES INDEX

TABLES INDEX

1. INTRODUCTION

1.1 Purpose

This document describes the main functions and how to use the Simcenter 3D HIGH FREQUENCY EMframework.

1.2 Application Field

The framework supports the user in all the phases of the typical work flow of antenna design and placement on complex platforms:

- antenna modelling and validation in a measurement set-up or a stand-alone configuration;
- antenna placement optimization with respect to service requirements (operative post-processing);

1.3 Trademarks

Simcenter 3D is registered trademark of the Siemens Product Lifecycle Management Software Inc.

Other product or brand names used in this manual are trademarks of their respective owners.

1.4 Reference

The applicable versions of the following documents are the ones officially released at the time of the emission of the present document**.**

1.4.1 Bibliography

[BD1] W. C. Gibson, The Method of Moments in Electromagnetics, Chapman & Hall/CRC, 2008.

[BD2] Rao, S. M., D. R. Wilton, and A. W. Glisson. “Electromagnetic scattering by surfaces of arbitrary shape.” IEEE Transactions on Antennas and Propagation, Vol. AP-30, No. 3, May 1982, pp. 409–418.

[BD3] S.U. Hwu, R.D. Wilton, S.M. Rao, "Electromagnetic scattering and radiation by arbitrary conducting wire/surface configurations", IEEE APS Int. Symp. Syracuse, 1988.

[BD4] W.C. Chew, J-M. Jin, E. Michielssen and J. Song, “Fast and efficient algorithms in computational electromagnetics”, Artech House, 2001.

[BD5] T. B. A. Senior and J. L. Volakis, “Approximate Boundary Conditions in Electromagnetics”, England: IEE, 1995.

[BD6] P. Yla-Oijala, M. Taskinen, and J. Sarvas, "Surface Integral Equation Method for General Composite Metallic and Dielectric Structures with Junctions," Progress In Electromagnetics Research, Vol. 52, 81-108, 2005.

[BD7] Vipiana, F., Pirinoli, P., and Vecchi, G., “A Multiresolution Method of Moments for Triangular Meshes,” IEEE Transactions on Antennas and Propagation 53(7):2247-2258, July 2005.

[BD8] De Vita P., Freni A., Vipiana F., Pirinoli P., Vecchi G. “Fast Analysis of Large Finite Arrays With a Combined Multiresolution-SM/AIM Approach” IEEE Trans. on Antennas and Propagation, Vol. 54, Iss.12, Dec. 2006 Pages:3827-3832.

[BD9] K.A.Michalski, J.R.Mosig, ”Multilayered Media Green’s Functions in Integral Equation Formulations”, IEEE Trans. Antennas Propagat. Vol.45, no.3, March 1997, 508-519

[BD10] G. Antonini, D. Romano, M. Bandinelli, A. Mori. A. Dieudonne Goleanu, M. Dunand, “A Surface PEEC Formulation for High-Fidelity Analysis of the Current Return Networks in Composite Aircrafts,” in *IEEE Transactions on Electromagnetic Compatibility*, vol. 57, no. 5, pp. 1027-1036, Oct. 2015.

[BD11] M. Bandinelli, A. Mori, G. Antonini, D. Romano and G. M. Sammarone, “Numerical analysis of avionic grounding structures with surface PEEC formulation”, 2015 9th European Conference on Antennas and Propagation (EuCAP), Lisbon, 2015, pp. 1-5.

[BD12] K. Zhao, M. N. Vouvakis, Jin-Fa Lee, “The Adaptive Cross Approximation Algorithm for Accelerated Method of Moments Computations of EMC Problems,” IEEE Trans. On Electromagnetic Compatibility, Vol. 47, No. 4, November 2005, pp. 763-773.

[BD13] M. Bebendorf, “Approximation of boundary element matrices,” Numer. Mathem., vol. 86, pp. 565–589, June 2000.

[BD14] M. Bandinelli, A. Mori, G. Antonini, D. Romano and G. M. Sammarone, "Numerical analysis of avionic grounding structures with surface PEEC formulation," 2015 9th European Conference on Antennas and Propagation (EuCAP), Lisbon, 2015

[BD15] C.W.I. Pistorius, D.A. Mc Namara J.A.G. Malherbe: "Introduction to the Uniform Geometrical Theory of Diffraction" , Artech House Boston - London, 1990

[BD16] R.C. Hansen, Ed., "Geometric Theory of Diffraction", IEEE Press, New York, 1981

[BD17] R. G. Kouyoumjian and P. H. Pathak, "A uniform geometrical theory of diffraction for an edge in a perfectly conducting surface," Proc. IEEE, Vol. 62, n.11, pp. 1448-1461, Nov. 1974.

[BD18] Sahalos, J. N., and G. A. Thiele, "A hybrid UTD - eigenfunction method for scattering by a vertex," ACES' Journal, 6 (1), 59-74, 1991.

[BD19] Hansen, T. B. "Corner diffraction coefficients for the quarter plane," IEEE Trans. Antennas Propagat., AP-39 (7), pp. 976-984, 1991.

[BD20] Sikta, F. A., W. Burnside, T. T. Chu, L. Peters Jr., "First-order equivalent current and corner diffraction scattering from flat plate structures," IEEE Trans. Antennas Propagat., AP-31 (4), pp.584-589, 1983.

[BD21] K. H. Hill, "A UTD solution to the EM scattering by the vertex of the scattering by polyhedral structures including higher terms," IEEE Trans. on Antennas Propagat., AP-40 (11), 1302-1312, 1992.

[BD22] S. Maci, R. Tiberio, A. Toccafondi "Diffraction at a plane angular sector", Jour. on e.m. Wave and Applic., vol.8, N. 9\10 pp. 1247-1276 September 1994

[BD23] F.Capolino, S. Maci "Simplified, Closed-Form Expressions for Computing the Generalized Fresnel Integral and Their Application to Vertex Diffraction" Microw. and Opt. Tech. Letters, vol. 9, n.1, pp. 32-37. May 1995.

[BD24] F. Capolino, S. Maci "Uniform high-frequency description of singly, doubly, and vertex diffracted rays for a plane angular sector" to be published on Jour. on e.m. Wave and Applic., 1996.

[BD25] Hansen J.E., Spherical Near-Field Antenna Measurements, Peter Peregrinus Ltd. London, 1988.

[BD26] F. Obelleiro, J. L. Rodriguez, and R. J. Burkholder, “An iterative physical optics approach for analyzing the electromagnetic scattering by large open-ended cavities,” IEEE Trans. Antennas Propag., vol. 43, no. 4, Apr. 1995, pp. 356–361.

[BD27] J. L. Rodriguez, F. Obelleiro, and A. G. Pino, “Iterative solutions of MFIE for computing electromagnetic scattering of large open-ended cavities,” IEE Proc.-Microw, Antennas Propag., vol. 144, no. 2, Apr. 1997 pp. 141–144.

[BD28] P. H. Pathak and R. J. Burkholder, “Modal, ray and beam techniques for analyzing the EM scattering by open-ended waveguide cavities,” IEEE Trans. Antennas Propag., vol. AP-37, no. 5, May 1989, pp. 635–647.

[BD29] F. Obelleiro, M. G. Araújo, and J. L. Rodriguez, “Iterative physical-optics formulation for analyzing large waveguides with lossy walls,” Micro. Opt. Tech. Lett., vol. 28, no. 1, Jan. 2001, pp. 21–26.

[BD30] R. J. Burkholder and T. Lundin, “Forward-backward iterative physical optics algorithm for computing the RCS of open-ended cavities,” IEEE Trans. Antennas Propag., vol. 53, no. 2, Feb. 2005, pp. 793–799.

[BD31] M. F. Cátedra, C. Delgado, and I. G. Diego, “New physical optics approach for an efficient treatment of multiple bounces in curved bodies defined by an impedance boundary condition,” IEEE Trans. Antennas Propag., vol. 56, no. 3, Mar. 2008, pp. 728–736.

[BD32] R. Hémon, P. Pouliguen, H. He, J. Saillard, and J. F. Damiens, “Computation of EM field scattered by an open-ended cavity and by a cavity under radome using the iterative physical optics,” Progress In Electromagnetics Research, vol. 80, 2008, pp. 77–105.

[BD33] R. J. Burkholder, Ç. Tokgöz, C. J. Reddy, and W. O. Coburn, “Iterative physical optics for radar scattering predictions,” Appl. Computational Electromagn. Soc. J., vol. 24, no. 2, Apr. 2009, pp. 241–258.

[BD34] P. Pouliguen, R. Hémon, and J. F. Damiens, “Physical optics for large scale electromagnetic scattering problems,” 2010 Proceedings of the Fourth European Conference on Antennas and Propagation (EuCAP), Barcelona, Spain, April 12–16, 2010.

[BD35] Saad, Y. (2003). “Iterative methods for sparse linear systems” Philadelphia, Pa. Society for Industrial and Applied Mathematics.

[BD36] R. J. Burkholder, “A fast and rapidly convergent iterative physical optics algorithm for computing the RCS of open-ended cavities,” Appl. Computational Electromagn. Soc. J., vol. 16, no. 1, Mar. 2001, pp. 53–60.

[BD37] A. Boag, “A fast iterative physical optics (FIPO) algorithm based on non-uniform polar grid interpolation,” Micro. Opt. Tech. Lett., vol. 35, no. 3, Nov. 2002, pp. 240–244.

[BD38] C. C. Lu and W. C. Chew, “Fast far-field approximation for calculating the RCS of large objects,” Micro. Opt. Tech. Lett., vol. 8, no. 5, Apr. 1995, pp. 238–241.

[BD39] L. Pandolfo, P. De Vita, M. Bandinelli, G. Carluccio, M. Albani, F. Vernì, “GPU accelerated iterative physical optics algorithm for the analysis of electrically large optics and platforms”, In 36th ESA Antenna Workshop on “Antennas and RF Systems for Space Science”, ESA/ESTEC, Noordwijk, The Netherlands, October 2015.

[BD40] G. Carluccio, M. Albani, L. Pandolfo, P. De Vita and M. Bandinelli, "Efficient Iterative Physical Optics algorithm for electrically large scatterers", Radio Science Meeting (Joint with AP-S Symposium), 2014 USNC-URSI, Memphis, TN, 2014, pp. 169-169.

[BD41] Chew W C, Cui T J, Song J M. “A FAFFA-MLFMA Algorithm for Electromagnetic Scattering.” IEEE Trans Antenna Propagat, 2002,50:1641–1649.

[BD42] Di Maria, L. Pandolfo, P. De Vita, G. Guida, M. Bercigli, M. Bandinelli, “High Fidelity Modeling of 30m Near-Field Cassegrain Antenna in X-band”, Proceedings of the Eighth European Conference on Antennas and Propagation (EuCAP), The Hague, The Netherlands, April 6–11, 2014.

[BD43] RTCA DO-213 Minimum Operational Performance Standards for Nose-Mounted Radomes, March 17, 2016.

[BD44] Balanis, C. A. (2005) “Antenna theory: Analysis and design” (3rd ed.). Hoboken, NJ: John Wiley.

[BD45] Collin, R.E. (1992) “Foundations for Microwave Engineering” (2nd ed.), Tata McGraw-Hill, New York.

[BD46] K. S. Kunz and R. J. Luebbers, “The Finite Difference Time Domain Method for Electromagnetics,” CRC Press, Boca Raton, 1993.

[BD47] K. J. Vinoy and R. M. Jha, “Radar Absorbing Materials: from Theory to Design and Characterization,” Kluwer Academic Publishers, Boston, 1996

[BD48] C.C. Chen, “Transmission of Microwave Through Perforated Flat Plates of Finite Thickness”, IEEE Transaction on Microwave Theory and Techniques, vol. MTT-21, No 1, pp. 1-6, Jan. 1973

[BD49] S.W. Lee, G. Zarrillo, C.L. Law, “Simple Formulas for Transmission Through Periodic Metal Grids or Plates”, IEEE Transaction on Antennas and Propagation, vol. AP-30, No 5, pp. 904-909, Sep. 1982

[BD50] G. Zarrillo, K. Aguiar, “Closed-Form Low Frequency Solution for Electromagnetic Waves Through a Frequency Selective Surface”, IEEE Transaction on Antennas and Propagation, vol. AP-35, No 12, pp. 1406-1417, Dec. 1987

1.5 Acronyms and Definitions

1.5.1 Acronyms

ACA	Adaptive Cross Approximation
BEM	Boundary Element Methods
BiCGStab	BI-Conjugate STABilized
CFIE	Combined Field Integral Equation
CTR	Compact Test Range
EFIE	Electric Field Integral Equation
EWIS	Electrical Wiring Interconnect System
FaFFA	Fast Far-Field Approximation
FGMRes	Flexible Generalized Minimal RESidual
GMResR	Generalized Minimal RESidual Recursive
GO	Geometrical Optics
GPU	Graphics Processing Units
GTD	Geometrical Theory of Diffraction
IBC	Impedance Boundary Condition
ILUT	Incomplete LU
IPO	Iterative Physical Optics
JMRES	Jacobi Minimal Residual
LU	Lower-Upper
MLFMA	Multilevel Fast Multipole Algorithm
MoM	Method of Moments
MPIE	Mixed Potential Integral Equation
MR	Multi-Resolution
PEC	Perfect Electric Conductor
PEEC	Partial Element Equivalent Circuits
PMCHWT	Poggio-Miller-Chang-Harrington-Wu-Tsai
PO	Physical Optics
PWL	Piece-Wise Linear
RAM	Random Access Memory / Radar Absorbent Material (depending on the context)
RWG	Rao Wilton Glisson
SM-AIM	Sparse Matrix Adaptive Integral Method
S-PEEC	Surface Partial Element Equivalent Circuits
SPLU	Sparse LU
SWE	Spherical Wave Expansion

TE	Transverse Electric
TM	Transverse Magnetic
UTD	Uniform Geometrical Theory of Diffraction
VSWR	Voltage Standing Wave Ratio

1.5.2 Definitions

Antenna radiation pattern (or antenna pattern): the antenna radiation pattern or antenna pattern can be defined as "a mathematical function or a graphical representation of the radiation properties of the antenna as a function of space coordinates." [BD44].

Conformal surface: a surface of any shape meshed with an "EM Field Probe" mesh type; this is used to calculate the near field on an arbitrary point cloud with respect to the regular scans. -A "3D Current Distribution" can be derived from the near field calculated above the points of the conformal scan; the equivalent current obtained will have the shape of the scan.

Full-wave port: a full-wave port (the so called '**Delta Gap'** [BD1]) is the way of access of the electrical/electromagnetic stimulus (excitation) to a system. In practice it is a differential voltage applied between the common edges of a set of triangles or the common nodes between two wires. Antenna terminals are usually "ports". is the load "Port Excitation".

Field regions (far field/near field concepts): given an antenna with a maximum overall dimension D (to be valid, D must also be large compared to the wavelength (λ)), three regions can be identified [BD44]:

1. **Reactive near-field** region; this is the "portion of the near-field region immediately surrounding the antenna wherein the reactive field predominates. For most antennas, the outer boundary of this region is commonly taken to exist at a distance $R < 0.62\sqrt{D^3/\lambda}$ from the antenna surface. For a very short dipole, or equivalent radiator, the outer boundary is commonly taken to exist at a distance λ/2π from the antenna surface."
2. **Radiating near-field (Fresnel)** region; this is the "region of the field of an antenna between the reactive near-field region and the far-field region in which radiation fields predominate and the angular field distribution depends on the distance from the antenna. If the antenna has a maximum dimension that is not large compared to the wavelength, this region may not exist. For an antenna focused at infinity, the radiating near-field region is sometimes referred to as the Fresnel region on the basis of analogy to optical terminology. […] The inner boundary is taken to be the distance $R < 0.62\sqrt{D^3/\lambda}$ and the outer boundary the distance $R < 2\,D^2/\lambda$. This criterion is based on a maximum phase error of π/8. In this region the field pattern is, in general, a function of the radial distance and the radial field component may be appreciable"
3. **Far-field (Fraunhofer)** region; this is the "region of the field of an antenna where the angular field distribution is essentially independent of the distance from the antenna". The far-field region is commonly taken to be at distances greater than

$2\,D^2/\lambda$ from the antenna. More specific rules can be found as a function of D, the maximum overall dimension of the antenna:

a. $D > 2.5\lambda \Rightarrow R < 2\,D^2/\lambda$
b. $\lambda/5 < D < 2.5\lambda \Rightarrow R > 5D$
c. $D < \lambda/5 \Rightarrow R > \lambda$

Free space: is the medium with neutral behaviour from the point of view of the EM phenomena (i.e. relative dielectric permittivity and relative magnetic permeability are respectively $\varepsilon_r = 1 + j0$ and $\mu_r = 1 + j0$); in most case it can be used to simulate "Air".

Poynting vector, S(θ, φ): this is used to describe the power associated instantaneous to an electromagnetic wave [BD44]. The instantaneous Poynting vector formula is:

$$\vec{S} = \vec{E} \times \vec{H}$$

where $\vec{E}$ and $\vec{H}$ are respectively the instantaneous electric and magnetic field vector.

Reference systems adopted: the Simcenter 3D High Frequency EM considers the following reference systems, depending on the far-field/near-field context.

- ***Far Field***
  - **Spherical coordinates system in far field (θ, φ)**: used to identify the directional coordinates of points in a sphere with ideally infinite radius where the EM far field is to be calculated.

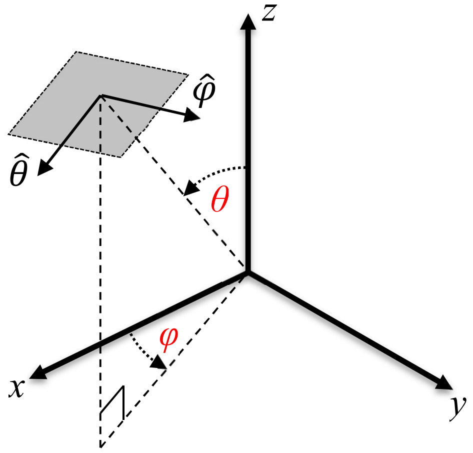


Fig. 1.1 – Spherical coordinates system in far field (θ, φ)

Valid values for θ are in the range (-180, 180) deg., while valid values for φ are in the range (-360, 360) deg.
Note: a correct definition of the geometrical scan range should avoid redundancy in the angular ranges.

  - **Direction cosines coordinates system (*u*, *v*)**: derived from the (θ, φ) coordinates to re-map the points where the EM far field is to be calculated in a rectangular space (U/V Space). The re-mapping formulas are:

$$u = \sin\theta\cos\varphi$$
$$v = \sin\theta\sin\varphi$$

with

$$-1 \leq u \leq 1$$
$$-1 \leq v \leq 1$$
$$u^2 + v^2 \leq 1$$

The UV space only re-maps the upper space hemisphere.

- ***Near Field***
  - **Cartesian coordinates system (x, y, z)**: this is the classical Cartesian coordinate system (Fig. 1.2).

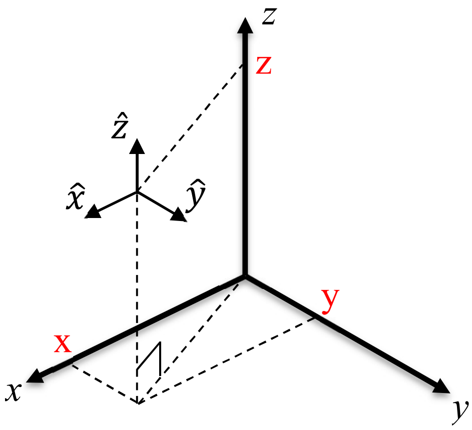


Fig. 1.2 – Cartesian reference system (x, y, z)

  - **Spherical coordinates system (ρ, θ, φ)**:

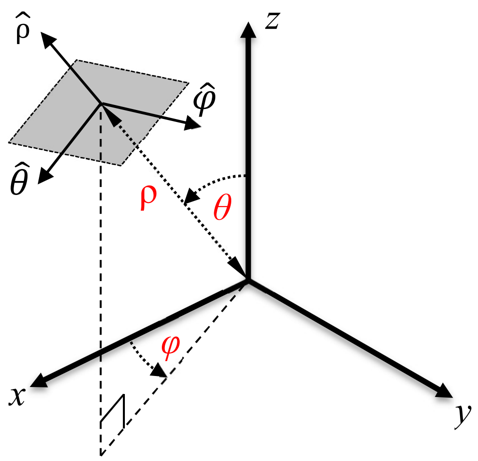


Fig. 1.3 – Spherical coordinates system (ρ, θ, φ)

Valid values for θ are in the range (-180, 180) degrees, while valid values for φ are in the range (-360, 360) degrees.
Note: a correct definition of the geometrical scan range should avoid redundancy in the angular ranges.

Synthetic source: all sources that do not need a mesh model to simulate the EM phenomena; these are mainly equivalent current distribution or antenna pattern models. They can be distinguished from full-wave (e.g. delta-gap) because they need a mesh to be applied. Synthetic sources include the following types of load:

- Electric/Magnetic dipoles (“Infinitesimal Dipole”)
- EM Plane Wave
- Pattern
- SWE
- 2D and 3D current distribution

Structure (or platform): this is intended as any object under analysis that is meshed to evaluate the EM effects on it: e.g. vehicles, aircrafts, ships, scenarios, etc.

2. OVERVIEW

2.1 Contents

This document describes the main functions available in the ***Simcenter 3D High Frequency EM*** and how to use them. The manual is divided into four main sections, as listed below:

- chapter 2 gives an overview of the overall product;
- chapter 3 details the minimum system requirement of the system;
- chapter 4 is the user guide;
- chapter 5 is the reference guide.

2.2 Intended readership

No pre-requisites are required for the ***Simcenter 3D High Frequency EM*** user.

Being a simulation environment for the analysis of electromagnetic problems, the User should have a basic knowledge of electromagnetism.

2.3 Basic functions

Simcenter 3D High Frequency EM provides support through the entire design cycle of complex platforms: from antenna modelling, through radiating systems performance optimization, up to system level analyses.
Streamlined working procedures ease and speed up the typical work flow of system integrators:

- identification of requirements (e.g. link requirements, standards, norms);
- antenna modelling and validation in a measurement set-up or a stand-alone configuration;
- antenna placement optimization with respect to service requirements (operative post-processing);

A straightforward repetition of the modelling process reduces the analysis time when many antennas are installed on a platform and many optimization loops have to be performed to optimize system level performance.
A palette of fully validated modelling methods (**multicode approach**) enables the analysis of complex platforms over the full operational frequency spectrum of modern aircrafts: from HF band, through V/UHF and GPS, up to RADAR band.

Special processing functions and graphical procedures to verify **system level performance** are available.

3. SOFTWARE INSTALLATION AND CONFIGURATION

The ***Simcenter 3D High Frequency EM*** product is made for a 32 bit or 64 bit computer running the Microsoft Windows operative systems.

The installation procedure consists of simply copying the directory containing the post-processor binaries in the desired location.

3.1 Minimum system requirements

64 bit Windows	4 GB of RAM; NVIDIA QUADRO graphical adapter, with 512 MB of RAM 1 GB available on Hard Disk as a minimum. Intel EM64T processors; Windows Seven® 64 bit, Windows 8.1® 64 bit, Windows 10® 64 bit. Microsoft Visual C++ 2010 Redistributable x64 Microsoft .NET Framework 4

Tab. 3.1 – Minimum system requirements

4. USER'S GUIDE

4.1 General Concepts

4.1.1 Materials management

In Simcenter 3D High Frequency EM, materials management is based on the "High Frequency" model material.

The **"High Frequency" model material** is used to have a high frequency range representation of all materials covering the entire range of EM characteristics from dielectric through to the good conductors.

Several mathematical models are available for a "High Frequency" model material [BD46]:

- Const+Sigma
- Debye 1st order
- Drude 1st order
- Lorentz
- Multiple Poles
- Multiple Frequencies

This model is used to define:

1. "EM Physical Properties" (4.1.2) material for:
   a. equivalent surfaces (2D mesh collectors)
   b. equivalent wires (1D mesh collectors)
2. the bulk material filling the "Volumetric EM Data" (4.1.3);
3. the "Stack Layer" (4.1.4, EM laminate modelling).

4.1.1.1 Const + sigma

The "Const + Sigma" model can be used to describe a material that has an EM behaviour characterised by constant values.

For the electric part, the model is:

- ε_r, the real part of ε
- ε_i, the imaginary part of ε
- σ_e, the electric conductivity

The dispersion relation is:

$$\varepsilon(\omega) = \varepsilon_r - j(\varepsilon_i + \frac{\sigma_e}{\omega\, \varepsilon_0})$$

Where ε is relative permittivity and σ_e the electric conductivity.

For the magnetic part, the model is:

- μ_r, the real part of μ;
- μ_i, the imaginary part of μ;
- σ_m, the magnetic conductivity;

The dispersion relation is:

$$\mu(\omega) = \mu_r - j(\mu_i + \frac{\sigma_m}{\omega\,\mu_0})$$

Where μ is relative permeability and σ_m the magnetic conductivity.

Note: if conductivity is equal zero, the model is not dispersive and the EM parameter is constant versus frequency.

4.1.1.2 Debye 1st order

The Debye model can be used to describe a material that has an EM behaviour characterised by a single first order pole. It is described by the following equation [BD46]:

$$\varepsilon(\omega) = \varepsilon_\infty + \frac{\varepsilon_s - \varepsilon_\infty}{1 + j\omega\, t_0}$$

Where:

- ε_∞ is the relative permittivity at infinite frequency;
- ε_s is the relative permittivity at zero frequency;
- t_0 is the relaxation time in seconds.

4.1.1.3 Drude 1st order

The Drude model can be used to describe a material that has an EM behaviour characterised by a single first order pole. It is described by the following equation [BD46]:

$$\varepsilon(\omega) = 1 + \frac{\omega_p^2}{\omega(jv_c - \omega)}$$

Where:

- ω_p is the radiant plasma frequency;
- v_c is the collision frequency.

4.1.1.4 Lorentz

The Lorentz model can be used to describe a material that has an EM behaviour characterised by a single second order pole. It is described by the following equation [BD46]:

$$\varepsilon(\omega) = \varepsilon_\infty + (\varepsilon_s - \varepsilon_\infty)\frac{\omega_p^2}{\omega_p^2 + 2j\omega\delta_p + \omega^2}$$

Where:

- ω_p is the resonant frequency;
- δ_p is the damping coefficient;
- ε_∞ is the relative permittivity at infinite frequency;
- ε_s is the relative permittivity at zero frequency.

4.1.1.5 Multiple Poles

Some materials have frequency EM behaviour that can't be described using a single first or second order pole. If the number of second order poles is only one, the Multiple Poles model corresponds with the previously described Lorentz model: the model is a combination of second order Lorentz poles [BD46].

$$\varepsilon(\omega) = \varepsilon_\infty + (\varepsilon_s - \varepsilon_\infty) \sum_{p=1}^{n} \frac{G_p \omega_p^2}{\omega_p^2 + 2j\omega\delta_p + \omega^2}, \sum_{p=1}^{n} G_p = 1$$

Where:

- ω_p is the resonant frequency;
- δ_p is the damping coefficient;
- ε_∞ is the relative permittivity at infinite frequency;
- ε_s is the relative permittivity at zero frequency;
- G_p the weight parameter, such as $\sum_{p=1}^{n} G_p = 1$.

4.1.1.6 Multiple Frequencies

These materials have frequency dependant dielectric permittivity.

They are useful when a measured data or materials described by other mathematical models needs to be imported. The complex number representing the relative permittivity must be inserted for each frequency.

4.1.2 EM Physical properties

The material information for the 2D Mesh model can be chosen from the following list:

- Perfect Conductor;
- Dielectric Boundary;
- Aperture;
- Multilayer;
- Half Space;
- Grid;
- Surface Impedance;
- Tabulated data;

The "Multilayer" and the "Half Space" properties are based on the "High Frequency" model material (4.1.1).

The 1D mesh is modelled by the wire radius and a material model that can be either a "Perfect Conductor" or a "High Frequency" model material.

4.1.2.1 Perfect Conductor

"Perfect Conductor" or "Perfect Electric Conductor" (PEC) is an idealized material exhibiting infinite electrical conductivity or, equivalently, resistivity equal to zero. This is an abstract concept that can be applied each time you need to simulate a good conductor (e.g. copper) in most EM high frequency applications; i.e. in those cases where a very low resistivity of the material can be neglected.

4.1.2.2 Dielectric Boundary

"Dielectric Boundary" is a fictitious material used to identify a surface on which the condition of continuity of the fields will be imposed.

It is only used with the 3D MoM code to identify a closed surface ("volume") filled by a dielectric material.

4.1.2.3 Aperture

"Aperture" is a fictitious material used to identify a hole in an infinite ground plane.

It is only used with the 2.5D MoM code, to identify an aperture on a "PEC layer with Aperture" defined in a "Stack Layer" definition.

4.1.2.4 Multilayer

A "Multilayer" is a stack of parallel slabs terminating in "Air" (i.e. free-space) or "PEC". Each multilayer has a total thickness given by the sum of the single slab thicknesses. Each "Multilayer" can contain a maximum of 16 layers.

Each slab has two parameters:

1) The constituting material (i.e. a "high frequency model" material);
2) Its thickness.

When the electromagnetic wave illuminates the material, its behaviour is obtained considering each slab infinite in the plane orthogonal to the thickness.

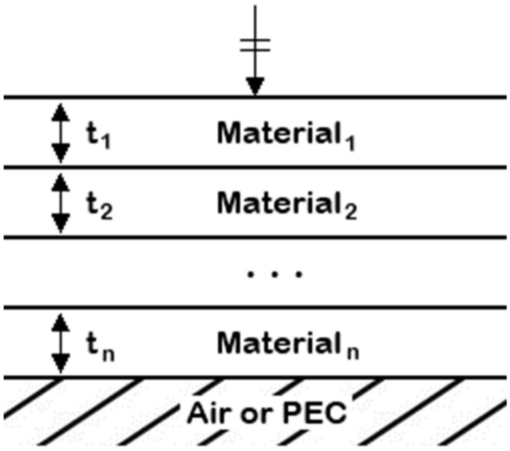


Fig. 4.1 – Multilayer representation

This kind of EM Physical Property is suitable for modelling dielectric slabs, resistive junctions, composite materials, Radar Absorbent Materials (RAM), etc.

4.1.2.5 Half Space

A “Half Space” material can be created with behaviour that is dependent on or independent of the frequency value using the “High Frequency” model material.

Note: when using Half Space, the transmission coefficient is equal to 0.

4.1.2.6 Grid

The “Grid” material consists of a PEC sheet with holes in it with a specific geometry [BD48], [BD49], [BD50].

A“Grid” material can be defined with the following parameters:

- The **Type of Hole** (Circular or Square);
- The **Grid Lattice** (Triangular or Square);
- The grid **Thickness**;
- The **Hole Diameter**;
- The **Center Spacing** (minimum distance between two hole centres).

Note: this material can only be used in a low frequency region, e.g. in a frequency band under the cut off frequency that is defined by the geometrical hole dimension.

4.1.2.7 Surface Impedance

The “Surface Impedance” representing a material is defined by inserting its “Real” and “Imaginary” parts.

4.1.2.8 Tabulated Data

"Tabulated Data" is defined by setting the perpendicular and parallel reflection and transmission coefficients (both real and imaginary parts) for the frequencies (up to 15) of interest.

For each frequency, you can define a specific value for different incidence angles, from 0° to 90°:

- **Angle\Coeff** is the incidence angle;
- **Re(R //)** is the real part of the reflection coefficient for the parallel polarisation;
- **Im(R //)** is the imaginary part of the reflection coefficient for the parallel polarisation;
- **Re(R ⊥)** is the real part of the reflection coefficient for the perpendicular polarisation;
- **Im(R ⊥)** is the imaginary part of the reflection coefficient for the perpendicular polarisation;
- **Re(T //)** is the real part of the transmission coefficient for the parallel polarisation;
- **Im(T //)** is the imaginary part of the transmission coefficient for the parallel polarisation;
- **Re(T ⊥)** is the real part of the transmission coefficient for the perpendicular polarisation;
- **Im(T ⊥)** is the imaginary part of the transmission coefficient for the perpendicular polarisation.

4.1.3 Volumetric EM Data

Only for 3D MoM analysis type.

Volumetric EM Data are associated to model mesh. Analysable geometries can consist of a mix of open and closed surfaces Each portion of space enclosed by a closed mesh shell is identified as a "volume". All model volumes and all the remaining space outside them, called "External Volume", make up the Volumetric EM Data for a given mesh model.

Materials can be associated with both surfaces and "volumes". Materials associated with surfaces (surface materials) are treated using approximate boundary conditions belonging to the category of impedance conditions [BD5]. A "volume" can be enclosed by a mesh made by a "dielectric boundary" or a surface material.

The "volumes" must be "filled" with homogeneous materials (bulk materials); in this case the solution of the problem is obtained without a priori approximations on the equations that describe the problem.

A "No field" option is available for each "volume"; this option defines a "volume" with no field inside it; this means that no sources are placed in the mesh to sustain the inner field, thus the RAM consumption is lower with respect to the case with field inside.

If the "default" Volumetric EM Data are used, all the "volumes" are filled by the "Free Space" material.

Full-wave Ports and Lumped Impedance (Regions) cannot be placed on the geometry/mesh in a way that divides two different volumes.

4.1.4 Stack Layer

Only for *2.5D MoM* analysis type.

When a planar radiant structure is embedded on a dielectric stack-up (e.g. an antenna), the geometrical (i.e. thickness) and EM characteristics of all dielectric materials (multi-layered structure properties) are provided using the *"*Stack Layer*"* physical properties.

It is important to point out that when using this kind of antenna, it is assumed that dielectric layers are infinite and homogeneous in the XY plane. This means that dielectric materials are defined by simply specifying their position and thickness in the layer stack-up.

The layer stack-up contains two layers by default:

- the "Lower Bound Layer", that describes the characteristic of the lower half space; this material is free space by default .
- the "Upper Bound Layer", that describes the characteristic of the upper half space; this material is free space by default.

Both "Lower" and "Upper" bound layers can be independently filled with a dielectric material or set as ground planes.

When simulating a metallic antenna in free space, the layer stack-up does not need to be modified.

You can specify the "Lower Bound Stack-up Reference Height": this defines the Z value where the "lower half space" ends; this is the height at which the first inserted "single layer" (the lower) will be placed. Adding more "single layers" means that they will be piled up in a stack, at the end of which the "upper half space" will start.

Each inserted layer can represent two different physical entities:

- a dielectric slab: in this case a filling material and a thickness must be defined; the material is represented by a "High Frequency" model;
- a "PEC Layer with Aperture": with this option enabled, the single layer is considered as an infinite ground plane with no thickness. This option is only useful when an "aperture" in the ideal ground plane is to be simulated. To do this, a mesh defining the aperture (i.e. the mesh will represent the "hole" in the ground) must be created at the same height as the "PEC Layer with Aperture"; in this case, the material associated to this mesh must be "Aperture" (4.1.2.3).

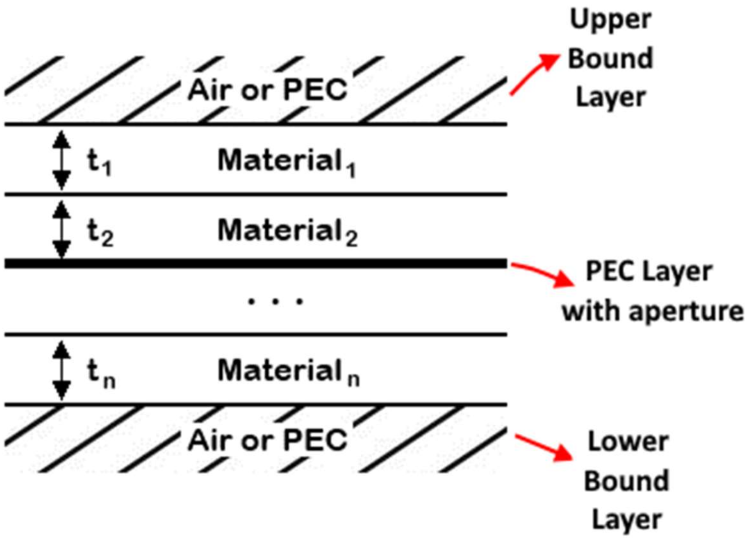


Fig. 4.2 – Stack Layer representation

4.1.5 Loads

4.1.5.1 Port Excitation

This is only available for a model to be solved using full-wave codes (3D MoM, 2.5D MoM and S-PEEC).

A "Port Excitation" (i.e. a full-wave port, the so called 'Delta Gap' [BD1]) is defined as the means of access of the electrical/electromagnetic stimulus (excitation) to a system. In practice, this is a differential voltage applied between the common edges of a set of triangles or the common nodes between two wires. Usually, antenna terminals are "ports".

4.1.5.2 Electric/Magnetic Dipole

The "Electric/Magnetic Dipole" is the "Infinitesimal Dipole" described in [BD44]. It has a uniform electric/magnetic current defined by a "Magnitude" and a "Length" that contributes to define the amplitude (i.e. amplitude is equal "Magnitude" × "Length"). The "Direction" is used to define the vector of the moment of the dipole located on a wire centred in the origin and located along the Z axis of the global reference system.

The position and orientation of the dipole reference system defines its installation point and orientation.

4.1.5.3 EM Plane Wave

The "EM Plane Wave" is a wave whose wave-fronts are infinite parallel planes perpendicular to the propagation direction $\hat{\beta}_0$, and whose peak-to-peak amplitude is constant. The propagation direction $\hat{\beta}_0$ is identified by two angles:

- θ is the angle between the $\hat{z}$ axis direction and the propagation direction, lying in the plane containing this two directions;
- φ is the angle between the $\hat{x}$ axis direction and the projection of the propagation direction in the XY-plane and lying on it.

The "Polarization Angle" of the wave is the angle η between the $-\hat{\theta}$ direction in the θ plane ($\hat{z}$ axis and propagation direction plane) and the electric field (E_0) lying in the plane orthogonal to the propagation direction (Fig. 4.3 b).

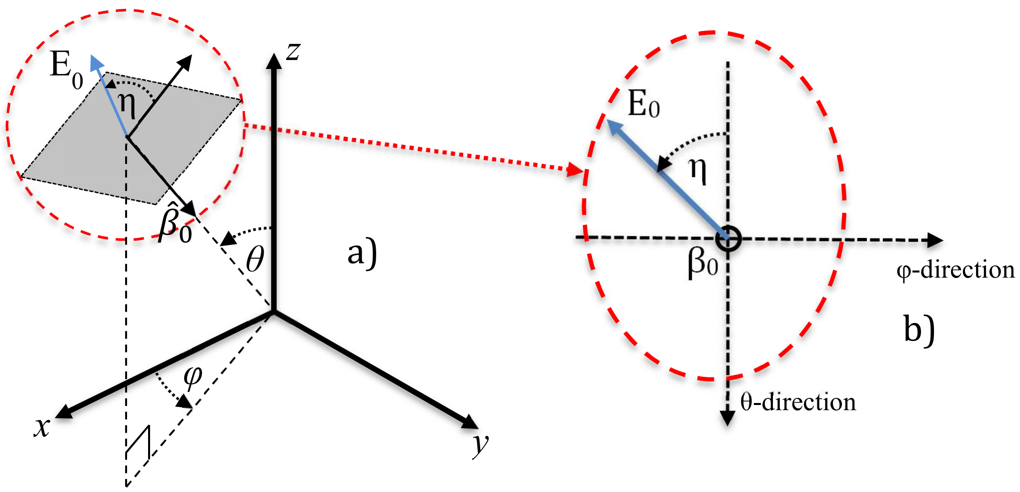


Fig. 4.3 – Representation of the incidence direction of the plane wave a) and its polarization angle (η) in the plane orthogonal to the propagation direction b)

4.1.5.4 Synthetic models: Pattern

The "Pattern" source model is simply the far field value radiated by the original source in each direction. It is well suited when the scattering structures and the observation point are located in the far field region (Fraunhofer's region).

4.1.5.5 Synthetic models: Spherical Wave Expansion (SWE)

The "Spherical Wave Expansion" (SWE) is a particular mathematical description of the radiated field that also enables field reconstruction in the near field region [BD25].

A SWE model is valid outside a sphere containing the antenna.

The radius (R_{min}) of this sphere is:

- $R_{min} \geq R$ if R is the radius of the smallest sphere containing the antenna;
- $R_{min} \geq \frac{\lambda}{4}\sqrt{\frac{D}{\pi}}$ if no information is available on the dimension of the antenna and D is the antenna directivity;

i.e. the radius of the sphere inscribed in the equivalent square aperture radiating with the given directivity D.

Our implementation uses “Q” type SWE .

Due to its nature, this kind of source is suitable for the description of antennas that are spatially limited, such as helix, cavity backed antennas, feeders, etc. with low gain and NOT for large antennas such as an array with high gain.

4.1.5.6 Synthetic models: 2D Current Distribution

This is an extended source model represented by a collection of elementary electric and magnetic sources (such as in the Huygens source representation) defined on a planar aperture.

4.1.5.7 Synthetic models: 3D Current Distribution

This is an extended source model represented by a collection of elementary electric and magnetic sources (RWG, PWL and attachments) defined on a 2D-triangular and 1D-wire mesh of an arbitrary surface in 3D space.

If the model is derived from a conformal near field, its equivalent currents will be $\hat{n} \times \bar{H}$ for the electric part and $\bar{E} \times \hat{n}$ for the magnetic part, placed in the same position as the mesh probe elements used to calculate the conformal near field.

4.1.6 Lumped Impedance

This is only available for models to be solved using full-wave codes (3D MoM, 2.5D MoM and S-PEEC).

It is a concentrated impedance connected between the common edges of a set of triangles or the common nodes between two segments. HFEM Regions are used to select the physical position of the lumped impedance; so it can be defined both at geometry level than at mesh level.

It can be represented as:

- RLC in series: model the electrical impedance by a resistance, a capacitance and an inductance in series;
- RLC in parallel: model the electrical impedance by a resistance, a capacitance and an inductance in parallel;
- Impedance (Z, in complex form): model the electrical impedance using an impedance;
- Admittance (Y, in complex form): model the electrical impedance using an admittance.
- External: the electrical impedance is loaded from an external Touchstone file (.s1p or .s2p; if .s2p is used, the second port will be considered to be short-circuited).

4.1.7 Outputs

4.1.7.1 SYZ-Parameters

This is only available for models to be solved using full-wave codes (3D MoM, 2.5D MoM and S-PEEC). Only full-wave ports are considered for this type of output.

The mesh model can be seen as the EM problem to be solved. For full-wave codes, this model can be represented as a black-box where the ports are the access modes of the electrical/electromagnetic stimulus (excitation) to the system. When the MoM Solution is solved, a “scattering matrix” consisting of the S-parameters is filled; each S-parameter is defined as $S_{mn} = \left.\frac{b_m}{a_n}\right|_{\substack{a_k=0 \\ k\neq n}}$ where b_m is the outgoing wave from the *m*-th port when in the n-th port a wave equal to a_n is entering in the condition of perfectly matched impendence at all the other ports apart from the *n*-th [BD45].

In a similar way, Y and Z-parameters can be defined and a conversion from one form to another can be calculated [BD45].

When the MoM Solution is solved, a matrix of parameters becomes available; only the full-wave ports will be taken in to account when calculating these parameters, synthetic sources do not affect them.

4.1.7.2 Impedance

This is only available for models to be solved using full-wave codes (3D MoM, 2.5D MoM and S-PEEC). Only full-wave ports are considered for this type of output.

The SYZ-Parameters are used to derive full-wave port impedance and other related quantities.

In particular, this output comprises [BD44]:

- Input impedance, Z [Ω]; calculated using the impedance of the generator of the ports;
- Input admittance, Y [S]; calculated using the impedance of the generator of the ports;
- Reflection coefficient, Γ; calculated using the reference impedance (default 50 Ω);
- Return Loss; calculated using the reference impedance (default 50 Ω);
- Voltage Standing Wave Ratio (VSWR); calculated using the reference impedance (default 50 Ω).

4.1.7.3 Induced Currents

This output represents the electric and/or magnetic currents that the electric stimulus (both full-wave ports, where applicable, and synthetic sources) induces on the mesh model.

The induced currents can be seen as the response of the structure to the electric stimulus.

They depend on the formulation used to solve the EM problem: in particular, the full-wave based codes currents could differ from the Iterative PO currents due to their different approach on the problem solving; moreover, the IPO currents are the base for all the other IPO outputs, while for the full-wave based codes they are only diagnostic information derived from the basis function contained in the MoM Solution.

4.1.7.4 Electric Potential

If the S-PEEC solver is used to analyse a structure, the user can evaluate the electric scalar potential generated on the structure for the set of excitation values set by the user. This corresponds to the electric scalar potential ϕ_e given in [BD44]. It can be used to estimate the electric potential difference between two points on the structures, but note that when by following the definition given in [BD44], the reference point is located at infinity.

4.1.7.5 Far Field

The far field output is the antenna radiation pattern (1.5.2) “determined in the far field region and is represented as a function of the directional coordinates. Radiation properties include power flux density, radiation intensity, field strength, directivity, phase or polarization.” [BD44].

This output needs the definition of a “Far Field Scan Area” containing the directions (space coordinates) where calculate the pattern. The definition could be in terms of (θ, φ) or in terms of (u, v) coordinates (1.5.2).

4.1.7.6 Near Field

The near field output is the antenna radiation behaviour determined at a finite distance and is represented as a function of point scans. Radiation properties include electric (E) and magnetic (H) field strength and phase, Poynting vector (S).

This output needs the definition of a “Near Field Scan Area” containing the points in the space where the near field will be calculated. The definition could be in terms of Cartesian coordinates (x, y, z), spherical coordinates (ρ, θ, φ) or a conformal surface (i.e. a meshed surface in a “mesh collector” of “EM Field probes”, 1.5.2).

4.1.7.7 Coupling

The coupling output evaluates the interaction between two sources (loads) placed together in the same analysis.

In practice, this output helps you to evaluate how antennas and any other kind of radiating apparatus influence each other.

High levels of coupling can lead to incorrect devices behaviour, affect services to be performed (e.g. radio air navigation systems, advanced driver assistance systems, etc.).

4.1.7.7.1 From S-Parameters

This is only available for models to be solved using full-wave codes (3D MoM, 2.5D MoM and S-PEEC). Only full-wave ports are considered for this output.

Using the S-Parameters, the calculation of the coupling between the model ports can be carried out; in this case four different types of inter-port coupling are evaluated:

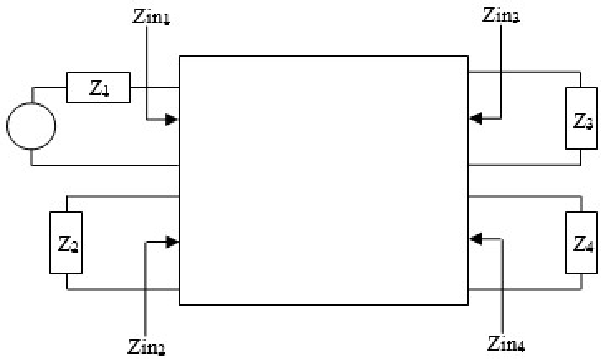


Fig. 4.4 – Multipole system representation for coupling purposes

- **Active Coupling,** i.e. coupling between the source port and the victim port, taking into account the real closing impedance conditions:
- **T Coupling,** Transducer Coupling, i.e. coupling taking into account the actual loading conditions for the output ports; no power reflection is taken into account at the input port.
- **A Coupling,** Available Input Power Coupling, i.e. coupling under the condition of maximum power transfer (Zj=Z*inj) for the output ports; impedance mismatch at the input port is taken into account.
- **Maximum Coupling**, i.e. inter-port coupling under the condition of maximum power transfer (Zj=Z*inj).

For definitions coming from amplifier power gain, see [BD45], par. 10.5.

4.1.7.7.2 Field-Field formulation

Using this formulation, coupling is evaluated by performing a “reaction integral” between two conformal near fields [BD44].

Note: the two near fields must be evaluated on the same geometrical scan (i.e. on the same points with the same accuracy level).

4.1.7.7.3 Field-Currents formulation

Available only for models to be solved using asymptotic codes (IPO and UTD), so only synthetic sources are foreseen.

Using this formulation, the coupling is evaluated relating the reaction (coupling) of fields radiated from the TX source to the RX source [BD44].

4.1.7.7.4 Friis formulation

Available only for models to be solved using asymptotic codes (IPO and UTD), so only synthetic sources are foreseen.

Using this formulation, the coupling is evaluated according to the Friis transmission formula, applying the formula to the single sources that make up each synthetic load; reflection and interaction with the platform are taken into account

Using the Friis transmission formula to evaluate the coupling implies that the RX source (the single constituting source for current distribution models) should be in the far-field region with respect to the structure and the TX source.

4.1.7.8 Equivalent Current Distribution

The equivalent current distribution output is a “3D Current Distribution” file to be used as input source for other solutions.

4.1.7.8.1 From MoM Solution

This is only available for models to be solved using full-wave codes (3D MoM, 2.5D MoM and S-PEEC).

Equivalent current distribution from MoM Solution is obtained from the conversion of the basis functions (RWG, PWL and attachments) that constitute the solution.

4.1.7.8.2 From Conformal Near Field

The equivalent current distribution from Conformal near field is obtained from the conversion of the near fields, $\hat{n} \times \bar{H}$ for the electric part and $\bar{E} \times \hat{n}$ for the magnetic part; the positions of the currents are the same as the “EM Field probes” elements used to calculate the conformal near field.

4.2 General Workflow

Step		Summary
1	Create a FEM and mesh the HF EM model.	Create a FEM using the **Simcenter 3D High Frequency EM Simulation** template. Set the Solver to **Simcenter 3D High Frequency EM** and the **Analysis Type** to **3D MoM**, **2.5D MoM**, **S-PEEC**, **Iterative PO** or **UTD**. Create the boundary mesh using the available 2D and 1D elements. Create the High Frequency EM material. For each mesh collector, choose the appropriate HF EM material and physical properties for your analysis. Create an EM Field Probe, if needed. Run the suitable checks.
2	Create a simulation	Create a simulation using **Simcenter 3D High Frequency EM** or the **Blank** Simulation template.
3	Create the solution	Create a new solution with **Solver** set to **Simcenter 3D High Frequency EM** and **Analysis Type** to **3D MoM**, **2.5D MoM**, **S-PEEC**, **Iterative PO** or **UTD**. Define the solution frequency range. Specify solution settings.
4	Create HF EM loads	HF EM loads may be **Full-wave port**, **Electric Dipole**, **Magnetic Dipole**, **EM Plane Wave**, **Synthetic models** such as **Pattern**, **Spherical Wave Expansion (SWE)**, **2D Current Distribution** or **3D Current Distribution**.
5	Create modelling objects	Create a **Frequency scan** to define a frequency range. Create a **Geometrical Scan Area** to define where the output must be evaluated, if needed.
6	Create simulation objects	Create an **Infinite Plane** to model an ideal ground reference in the XY-Plane at z = 0, if needed. Create a **Lumped Impedance** to add concentrated impedance to the model, if needed/applicable.
7	Select solution parameters	Select loads and simulation objects for the solution.
8	Solve the solution	Solve the solution to generate the analysis results.
9	Post-process results	Use post-processing tools, such as contour plots or XY graphing to analyse your results. Use HF EM Post-processing tools to display your output. Display **S/Y/Z-Parameters**, **Induced Currents**, **Far Field**, **Near Field**, **Coupling**, **Impedance**, **3D Current Distribution**.

4.3 3D MoM Analysis

4.3.1 General Concepts

The 3D MoM solver is a general purpose frequency domain Method of Moment [BD1] numerical code, based on the MPIE (Mixed Potential Integral Equation) for the modelling of antennas in stand-alone and installed on platform.

The method, belonging to the category of boundary element methods (BEM), is based on a sub-domain discretization (i.e. mesh) of the antenna/platform geometry (the Piece-Wise Linear – PWL on wire [BD1], the Rao Wilton Glisson – RWG on triangles [BD1] [BD2] and junctions between them [BD3] are implemented as basis functions), that permit the modelling of a large class of structures (i.e. no shape limitations).

Among the main features:

- a “multi-port” approach is adopted to evaluate ‘active’ observables (i.e. pattern, near-field, coupling) quickly for any kind of excitation configuration;
- the following materials can be managed: perfectly conductive materials, materials with losses and dielectrics materials;
- calculation can be performed with single or double precision variables;
- R, L, C, Y, Z lumped impedances;
- ideal ground plane is used;
- the geometry of the problem is represented “by nested and contiguous enclosed volumes” that are calculated and separated by a boundary on which apertures and openings can be defined (“Volumetric EM Data”);
- special algorithms enable the effective and accurate management of multi-scale models (i.e. models in which the ratio between the size of the largest and smallest mesh element is very high) and also render the solution stable for very low frequencies (i.e. frequency → 0 Hz);
- the solver is equipped with the Multilevel Fast Multipole Algorithm (MLFMA) to reduce computational costs [BD4];

Through the Method of Moments, the integral equations defined on the boundaries (i.e. the surfaces that describe the geometry of the problem) are discretized, obtaining a linear system characterized by a full matrix. This system can be solved either directly (standard MoM) or using the fast method MLFMA.

Analyzable geometries can be composed of a mix of open and closed surfaces. Closed surfaces identify “volumes”. The materials can be associated with both surfaces and “volumes”. The materials associated with the surfaces (surface materials) are treated using approximate boundary conditions belonging to the category of impedance conditions [BD5]. The “volumes” must be “filled” with homogeneous materials (bulk materials); in this case the solution of the problem is obtained without a priori approximations on the equations that describe the problem. A valid criterion for deciding whether to use surface materials (which reduce the computational cost of simulation) or bulk materials is related to thickness. If t is the thickness and $k_0 \cdot t \ll 1$ (where $k_0 = 2\pi/\lambda_0$ being λ_0 the wavelength in the vacuum) then the impedance boundary condition is a good approximation.

A special type of material attributable to a closed surface is the so called “dielectric boundary”, a fictitious material used to identify a surface on which the condition of continuity of the fields will be imposed.

A volume can be enclosed by a dielectric boundary or a surface material.

The 3D MoM uses different boundary conditions depending on the type of surface (i.e. material) on which these are applied:

- Electric Field Integral Equation (EFIE) on perfect conductor materials → a single set of basis functions (RWG) will be defined on a triangle if the relevant surface is open and a double set (one for each sides of the triangle) if the surface is the boundary of a volume. In any case, these functions will support electric equivalent currents.
- Impedance Boundary Conditions (IBC) on materials with loss → a double set of basis functions (RWG) will be defined on a triangle whether it is part of an open surface or of a closed surface. In the first case the functions will support electric and magnetic equivalent currents and in the second only electric currents (one for each sides of the triangle, magnetic currents will be implicit and depend linearly on electrical currents).
- Combined Field Integral Equation (CFIE) [BD1], if requested by the user and as an alternative to the EFIE, on perfect conductor or lossy materials if the relevant surface is closed → a double set of basis functions (RWG) will be defined on a triangle (for lossy materials, magnetic currents will be implicit and depend linearly on electrical currents).
- Poggio-Miller-Chang-Harrington-Wu-Tsai PMCHWT [BD6] on dielectric boundary → a double set of basis functions (RWG) will be defined on a triangle. The first set will support electric currents and the second one magnetic currents.

The solver automatically applies, according to the surface/material type, the most suitable boundary condition. On conductive surfaces, using the MLFMA, the user can choose whether to apply the EFIE/IBC condition (default) or the CFIE/IBC.

The types of material representation managed by the 3D MoM code are:

- Perfect Electric Conductor (4.1.2.1)
- Dielectric Boundary (4.1.2.2)
- Other Physical Materials
    - Multilayer (4.1.2.4)
    - Grid (4.1.2.6)
    - Surface Impedance (4.1.2.7): this material is considered by the 3D MoM code as an "opaque" material characterized by only a Z_S matrix, without the Z_T one.
    - Tabulated Data (4.1.2.8)

The types of sources managed by the 3D MoM code are:

- Full-wave port (4.1.5.1) (i.e. the so called 'Delta Gap' [BD1]);
- Elementary (electric and magnetic (4.1.5.2) small dipoles, plane waves (4.1.5.3));
- Equivalent models (currents based (4.1.5.6, 4.1.5.7) or SWE (4.1.5.5)).

  <u>Note</u>: full-wave ports (and in the same way, lumped loads) cannot lie on surfaces that are an interface between two "volumes".

3D MoM is able to calculate the following output (observables):

- Far Field (4.1.7.5);
- Near Field (4.1.7.6);
- Induced Currents (4.1.7.3);
- S-parameters (4.1.7.1), Coupling (4.1.7.7);
- Input Impedance/Admittance, Return Loss, Reflection Coefficient, VSWR (4.1.7.2);

It can also generate equivalent models (currents based) that can be used as sources in all other available analyses (4.1.7.8).

These outputs are calculated starting from the solution (*MoM Solution – S-Parameters*), in terms of induced currents, of the linear system that describes the problem. This is typically the most expensive part of the calculation. If the standard MoM is used, the solution will be direct (LU factorization) while the MLFMA requires an iterative solution (this is a typical feature of fast methods).

The computational cost of the iterative solution is linearly linked to the number of iterations which must therefore be as low as possible. This can be achieved by using a preconditioner, a transformation of the linear system capable of regularizing the problem by reducing its conditioning (in fact, the lower this parameter, the better the convergence will be).

The preconditioners can be "algebraic" (i.e. SPLU and ILUT, applicable in general regardless of the physical problem being faced) or "physical" (i.e. MR) who instead exploit the properties of the equations that are being solved. The former typically have a more general applicability but a higher computational cost for the generation while the latter have a lower computational cost.

The MoM based codes (3D but also 2.5D, see 4.4) are equipped with both these preconditioners. The physical preconditioner available is based on the Multi-Resolution (MR) technique [BD7] which is particularly effective in the case of multiscale models or characterized by a very dense mesh with respect to the wavelength (λ). If algebraic preconditioners are only available for the MLFMA, MR can also be used with the standard MoM to stabilize the linear system at low frequency.

For further details see Fig. 4.5.

4.3.2 Application Field

The Method of Moments is particularly indicated when the models are complex (in terms of geometries and/or materials) and the focus is on result accuracy. The algorithm is part of the family of so-called "full-wave" methods that solve the equations by describing the physics of the problem (i.e. Maxwell equations) in a discrete form, and not introducing other approximations (excluding lossy materials modelled using impedance boundary conditions) other than those due to the finiteness of the mathematics used for computer implementation. The ability to produce accurate results has its counterpart in the computational burden of the method. Given a model mesh with N_e triangles, N_w segments and N_a attachments (i.e. connections between segments and triangle vertexes), the number of the basis functions *N* will be

$$N \approx 1.5 \times N_e + N_w + N_a$$

in the case of a single set of functions per triangle and

$$N \approx 3 \times N_e + N_w + N_a$$

in the case of a double set of functions per triangle.

If the problem is discretized with *N* basis functions, the standard algorithm (i.e. not accelerated) will have a computational cost proportional to N^2 as regards the occupation of RAM and to N^3 for the calculation time. The use of the fast MLFMA method can considerably reduce the computational requirements up to the theoretical limit of $N\text{Log}N$ (both in terms of memory and calculation times). In any case, taking into account that the number of basis functions necessary to discretize a surface increases in a quadratic manner as the frequency of analysis increases, it is clear that once the hardware resources have been set, there will always be a maximum limit to the size of the geometries that can be handled.

Calculation of computational requests in terms of RAM in the case of the standard MoM is quite simple and reliable

$$\text{RAM} \approx N^2 \cdot \text{kvar [Bytes]}$$

where kvar=8 for single precision variables and kvar=16 for double.

Performing the simulation in single precision allows computational requests in terms of RAM to be halved (both for the standard MoM and for the MLFMA). In many cases, this is the best choice because typically no consequences on the accuracy of the results can be appreciated. Double precision is mandatory for low frequency analyses and in general when there are ill-conditioning problems in the MoM linear system. In this sense, benefits are possible even when there are convergence problems with the MLFMA. Another condition where it may be necessary to use double precision is for models in which the ratio between the maximum model size and mesh step is very large ($\sim 5 \cdot 10^3$).

When should we use standard MoM or MLFMA?

Standard MoM is mainly useful for low frequency analyses (electrical dimensions of the model less than few wavelengths) where it's very important to use the MR preconditioner. Another case where it may be convenient is for small models (N~<20,000) with many (>~10) sources. In all other cases MLFMA is the best choice.

For problems with closed surfaces (or at least with more closed surfaces than open surfaces), when there are no thin metallized planar substrates modelled as solid materials, the application of CFIE can significantly improve the convergence of the iterative method used to solve the problem with the MLFMA.

For very low frequency analyses, when the discretization step is lower than about $\lambda/10^6$, the use of a MR preconditioner may not be sufficient to avoid ill-conditioning problems. In these cases it's necessary to use the *low frequency stabilization* that, by separately processing the scalar and the vector potential, ensure the simulation frequency of a few ten hertz is reached.

4.3.3 Modelling rules

Great attention must be paid to generating a mesh of the geometry that is sufficiently dense to allow the evaluation of the observables of interest with the desired degree of accuracy.

The general rule is to use a discretization step which is about **one tenth of the wavelength of the surrounding medium**.

However, it may be useful to modify this rule in some cases:

- when you are interested in quantities that depend on the local solution of a problem (i.e. S-parameters, input impedance/admittance, return loss, reflection coefficient, VSWR); in these cases, it’s important to discretize these areas of the model with a step that is sufficient to take into account all the most important physical phenomena;

- when the geometric details are small but have a potentially important role in determining the solution; in these cases, it is necessary to use a sufficiently small discretization step to correctly describe the geometry;
- when, on the other hand, the observables of interest depend on the contribution of the currents on the whole model (far-field), we can use a larger discretization, but remains never less than one sixth of the wavelength of the surrounding medium.

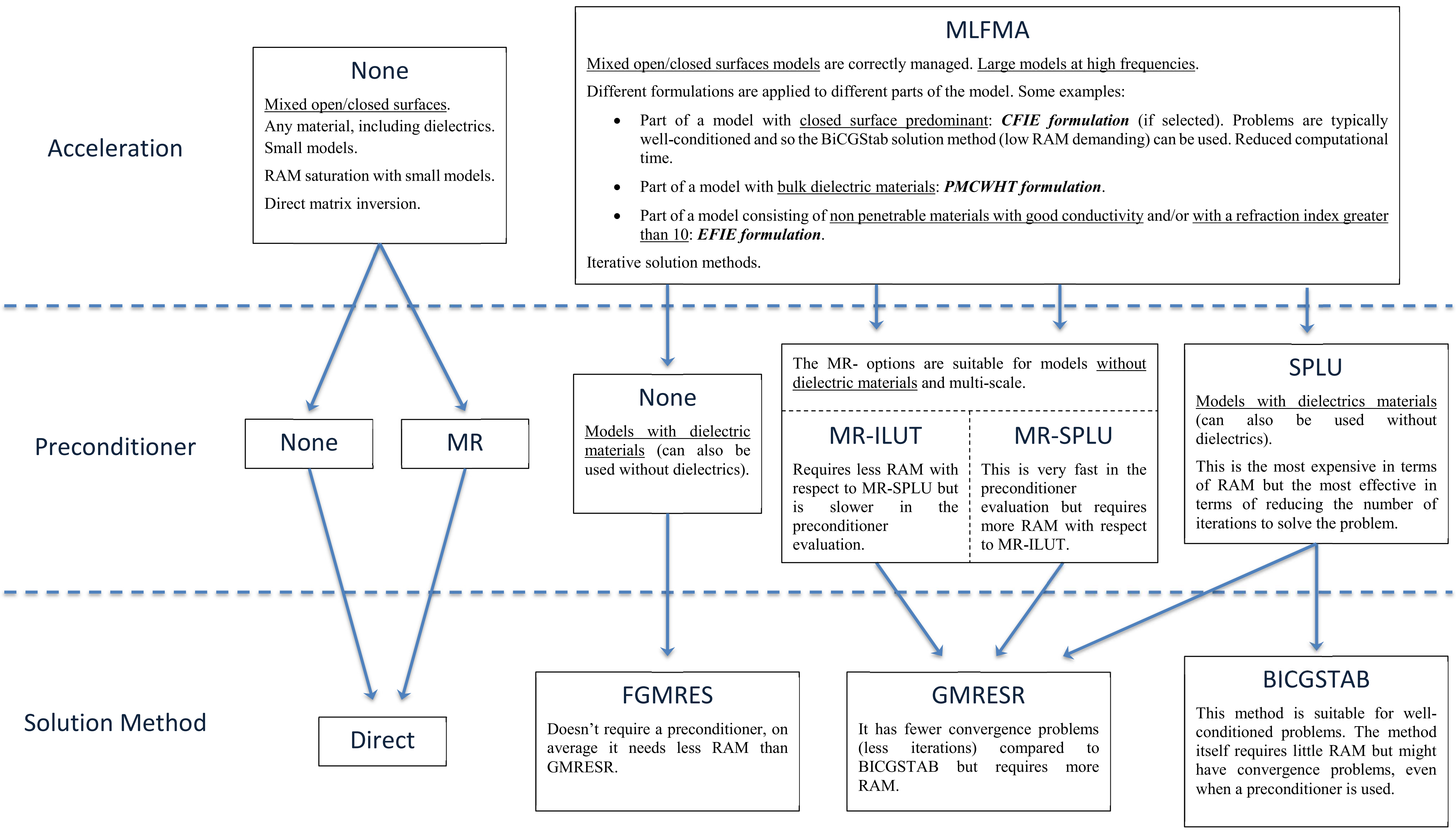


Fig. 4.5 – 3D MoM suggested settings

4.3.4 Workflow

Step		Summary
1	Create a FEM and mesh the HF EM model.	Create a FEM using the **Simcenter 3D High Frequency EM Simulation** template. Set the Solver to **Simcenter 3D High Frequency EM** and the **Analysis Type** to **3D MoM**. Use the **Stich Edge** function over the whole model. Insert a **Mesh Point** at the end of each wire where there is a connection with a patch (best results with **Type→Project Point**, **Projection Method→Closest Point**). Create the boundary mesh using the available **2D Triangle** and **1D Wire** elements. Create **High Frequency EM material**. For each mesh collector, choose the appropriate HF EM material and physical properties for your analysis. Create an **EM Field Probe** meshing the relevant part of the model, if needed. Run the appropriate checks, in particular **Duplicate Nodes** and **Duplicate Elements**.
2	Create a simulation	Create a simulation using **Simcenter 3D High Frequency EM** or the **Blank** Simulation template.
3	Create a MoM Solution	Create a new solution with **Solver** set to **Simcenter 3D High Frequency EM** **Analysis Type** set to **3D MoM**. **Solution Type** set to **MoM Solution – SYZ-Parameters** Specify solution settings.
4	Create HF EM loads	HF EM loads may be **Full-wave port**, **Electric Dipole**, **Magnetic Dipole**, **EM Plane Wave**, **Synthetic models** such as **Spherical Wave Expansion (SWE)**, **2D Current Distribution** or **3D Current Distribution**.
5	Create physical properties	Create **Volumetric EM Data** physical properties using the relevant button on the **HF EM functions** ribbon.
6	Create modeling objects	Create a **Frequency scan** to define the solution frequency range. Create a **Geometrical Scan Area** to define where the output must be evaluated, if needed.
7	Create simulation objects	Create an **Infinite Plane** to model an ideal ground reference in the XY-Plane at z = 0, if needed. Create a **Lumped Impedance** to add concentrated impedance to the model, if needed.
8	Select solution parameters	Select loads, physical properties, modeling objects and simulation objects for the solution.

9	Solve the solution	Solve the solution to generate the analysis results.
10	Post-process MoM Solution results	View the **S/Y/Z-Parameters** using the HF EM Post-processing tools.
11	Create the solution	Create a new solution with **Solver** set to **Simcenter 3D High Frequency EM** **Analysis Type** set to **3D MoM**. **Solution Type** set to **Induced Currents**, **Far Field**, **Near Field**, **Coupling**, **Impedance** or **3D Current Distribution**. Select the previously created **MoM Solution** to be used as a prerequisite. Create a **Geometrical Scan Area** to define where the output must be evaluated, if needed. Specify the solution settings.
12	Select solution parameters	Select loads and simulation objects for the solution.
13	Solve the solution	Solve the solution to generate the analysis results.
14	Post-process results	View the **Induced Currents**, **Far Field**, **Near Field**, **Coupling**, **Impedance** or **3D Current Distribution** using the HF EM Post-processing tools.

Tab. 4-1 – 3D MoM workflow

4.3.5 Application examples

4.3.5.1 Evaluation of the effect of the bumper on the radiating performances of an automotive radar

The problem is to assure good/controlled RF radar performance when installed aboard the car. The working frequency is 77GHz.

The radar consists of the radiating part (i.e. the antenna) and the control/processing electronics. Interaction with the car has the greatest impact on the performance of the radiant part, and this is what we will discuss below.

The problem can be tackled by subdividing it into two sub-problems:

1) antenna modelling;
2) antenna installation aboard the car.

Assuming that all the necessary information about the antenna (geometry and materials) is available, we can proceed with the creation of the full-wave model. Otherwise, an equivalent model must be used from the beginning: this will typically be a current based model that will have been generated starting from the knowledge of the far field radiation diagram or, in the most fortunate cases, from near field information.

Suppose we are in the first case, therefore, we have all the information needed to make a full-wave model.

Below is sketched the CAD model (Fig. 4.6). The antenna, a 3x2 array on a dielectric substrate (ε_r=3), is enclosed within a plastic protection box.

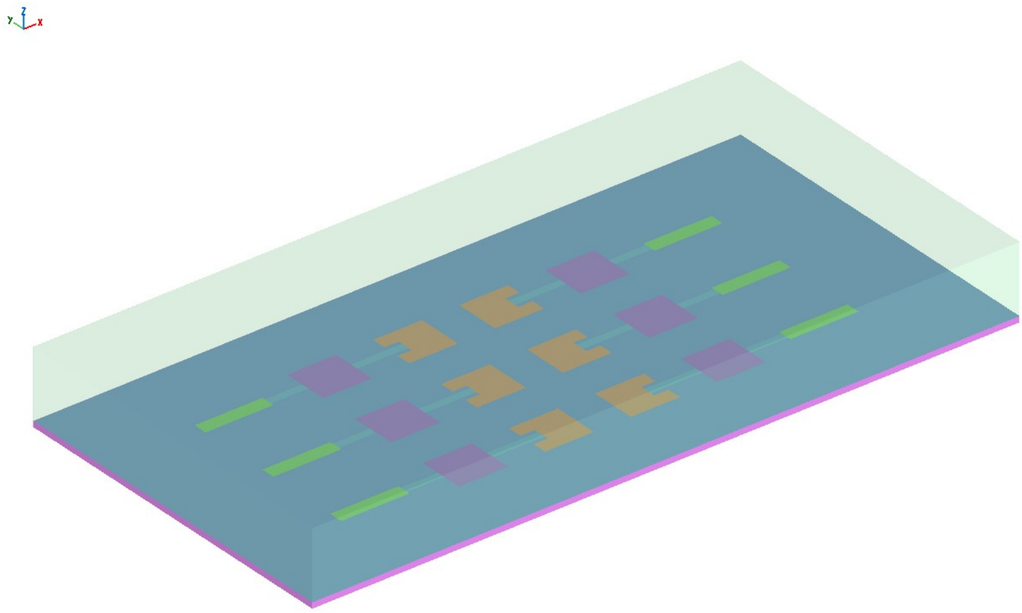


Fig. 4.6 – CAD model of the radar antenna

The first step is to create the mesh model. The discretization step can be chosen as a tenth of the wavelength ($\lambda = \frac{v}{f}$ where $v = \frac{1}{\sqrt{\varepsilon\mu}}$ and f is the frequency) of the surrounding medium.

For surfaces that overlook different materials (e.g. dielectric substrate and air), the average wavelength in the two media can be chosen as the reference wavelength for the mesh step.

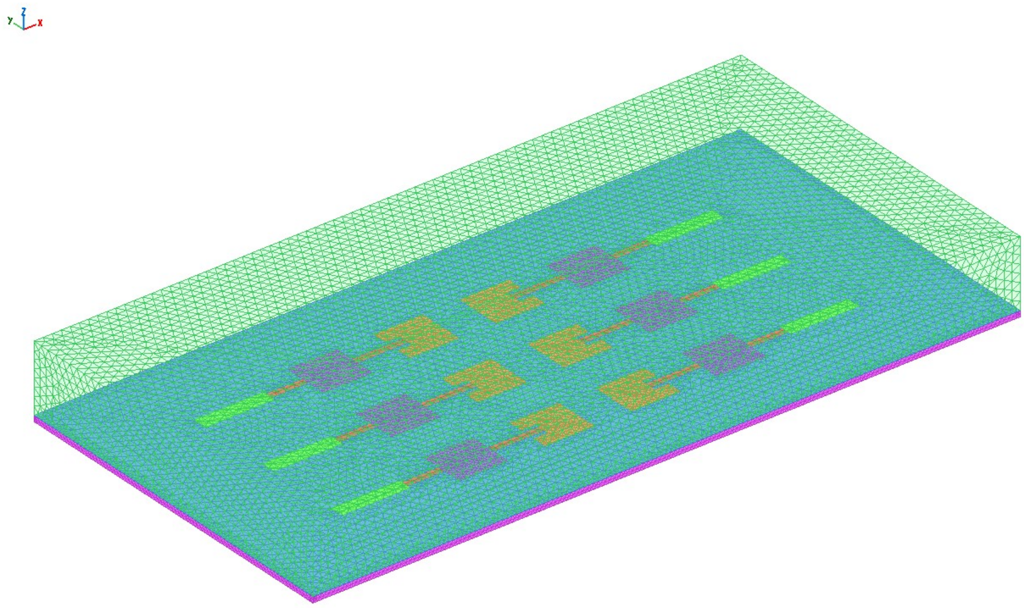


Fig. 4.7 – Mesh model of the radar antenna

To correctly model the main physical phenomenon, ensure that at least three rows of triangles are present on the feeding microstrips. For the same reason it is best to use a denser mesh (~λ/20) for the radiating part. The following picture (Fig. 4.8) shows the obtained mesh (consisting of about 40,000 triangles).

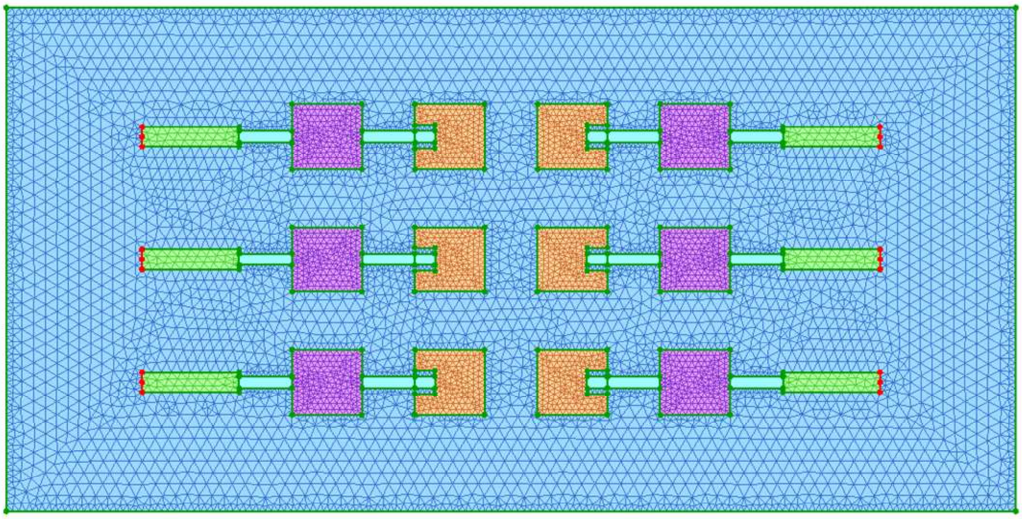


Fig. 4.8 – Mesh model detail of the radar antenna

The next step is to place the feeding ports (6 for this case, one for each array element). The ports (and in the same way, lumped loads) cannot lie on surfaces that are an interface between two volumes (see Fig. 4.9 for an example of correct placement).

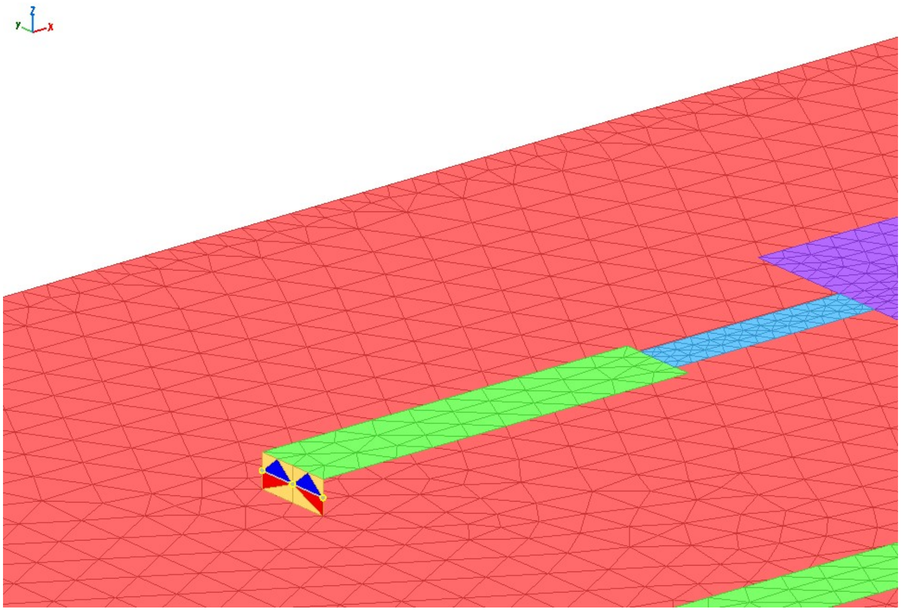


Fig. 4.9 – Full-wave port placement

Finally, the model generation requires you to assign the types of materials. In this case, four materials will be used. Three "surface materials":

- perfect conductor (PEC) for metal parts;
- dielectric boundary for all direct interface surfaces between the substrate and air;
- a plastic material (polyethylene based) for the protection box.

and a dielectric (ε_r=3) "bulk material" within the volume defined by the substrate.

Running the solution requires you to make some choices:

- whether to use the fast MLFMA method or not;
- to run the solution in single or double precision;

- whether to use the CFIE/IBC boundary condition or keep the default one (EFIE/IBC);

Given the number of mesh elements that make up the model and the fact that discretization is fairly regular (not multi-scale) and not particularly dense, it is advisable in our case to opt for single-precision MLFMA. The fact, however, that there are metallizations lying on a thin substrate makes it important to use the EFIE/IBC default boundary condition.

To speed up the convergence of the MLFMA, a preconditioner can be used. For medium/small models like this, the algebraic preconditioner SPLU is almost always very effective.

The execution of the solution and the calculation of the induced currents complete the first sub-problem (antenna modeling). Fig. 4.10 below shows the induced currents and the antenna pattern.

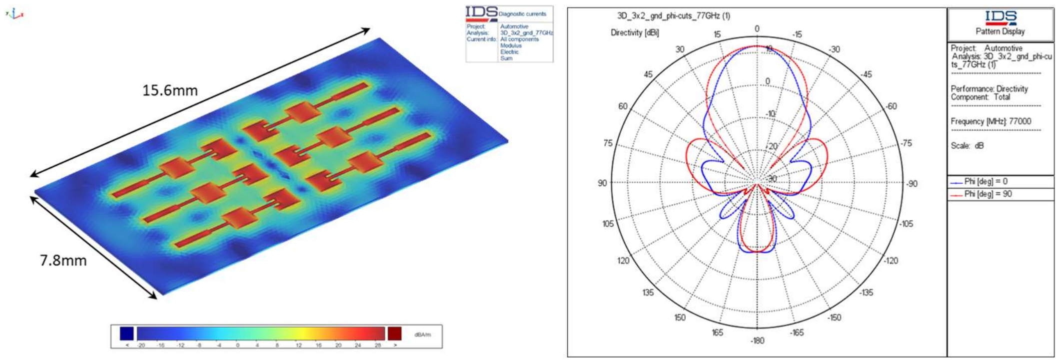


Fig. 4.10 – Induced currents and far-field pattern of the radar antenna

Let's consider the following antenna and bumper installation configuration (Fig. 4.11).

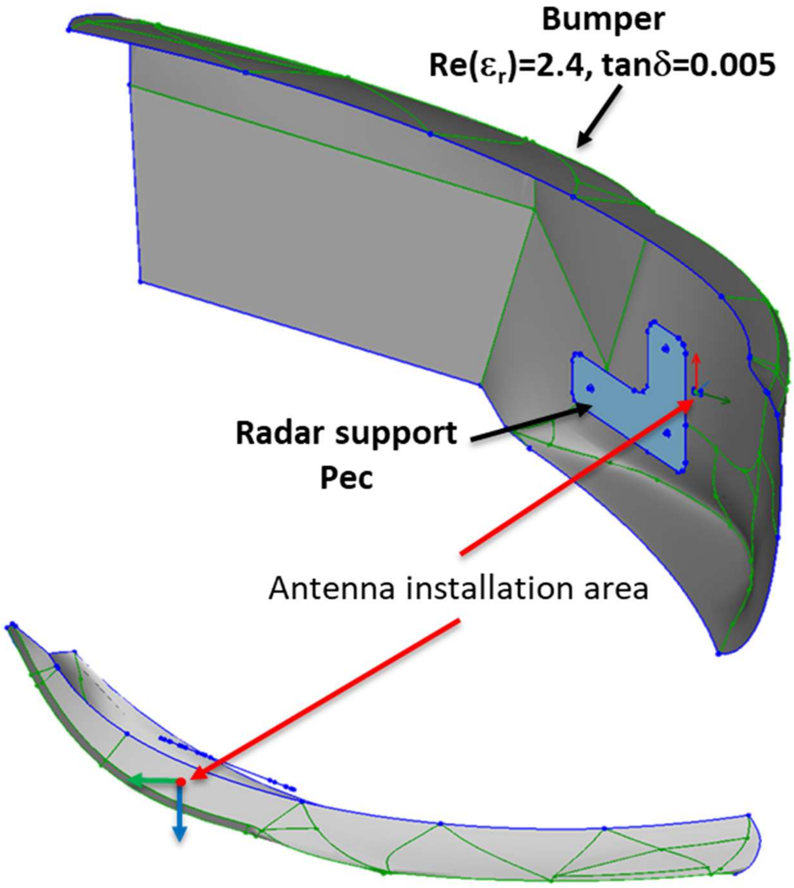


Fig. 4.11 – Radar antenna installation

Analysis of the effect of installing the antenna on the car can be addressed in one of two ways:

1) using an equivalent antenna model generated from induced currents installed behind the bumper (the latter modeled as full-wave);
2) through a full-wave solution of the entire problem (antenna + bumper).

To assess the effects of pattern distortion it's sufficient to use an equivalent model (current based), this approach reduces the computational cost.

On the contrary, if you are interested in the effect on the antenna port parameters (e.g. S-parameters) you need to opt for the more expensive full-wave analysis of the entire problem.

Here we will use an equivalent model approach.

The e.m. model of the bumper must be made. This consists of a shaped 'sheet' of plastic material with a thickness of 3mm,. Since $k_0 \cdot t > 4$, bulk modeling of the bumper material is required.

The part of the bumper that has the greatest effect on antenna performance is the area immediately in front of the radar. To reduce the computational cost of the solution, a subset of the bumper can be considered in the simulation.

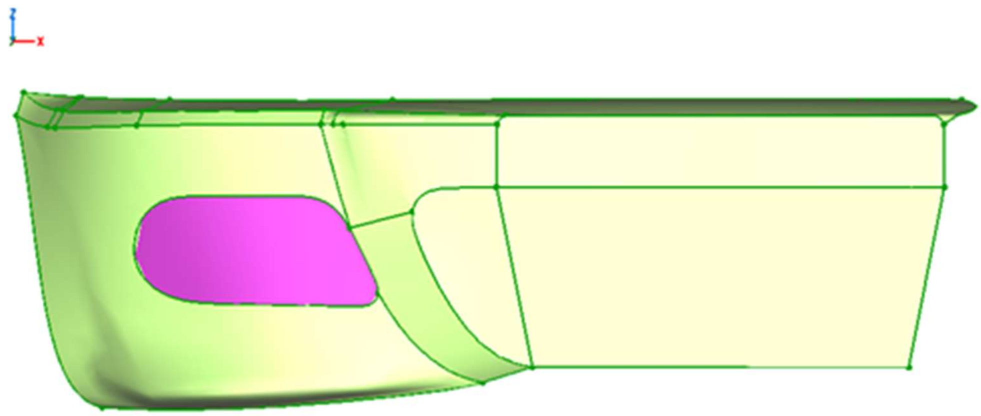


Fig. 4.12 – Bumper subset used for the simulation

First we make the mesh then we assign the materials:

- dielectric boundary to the triangles that define the surface of separation between the bumper plastic and air;
- PEC to the radar metallic support;
- dielectric (Re(ε_r)=2.4, tanδ=0.005) within the volume identified by the surface.

After antenna placement, we can proceed with the simulation: MLFMA in double precision will be used (due to the model dimension and to the high ratio between maximum bumper size and mesh step). To speed-up the convergence, the FGMRes iterative method will be selected. The following image (Fig. 4.13) shows a comparison between the patterns of the free-space stand-alone and installed antenna.

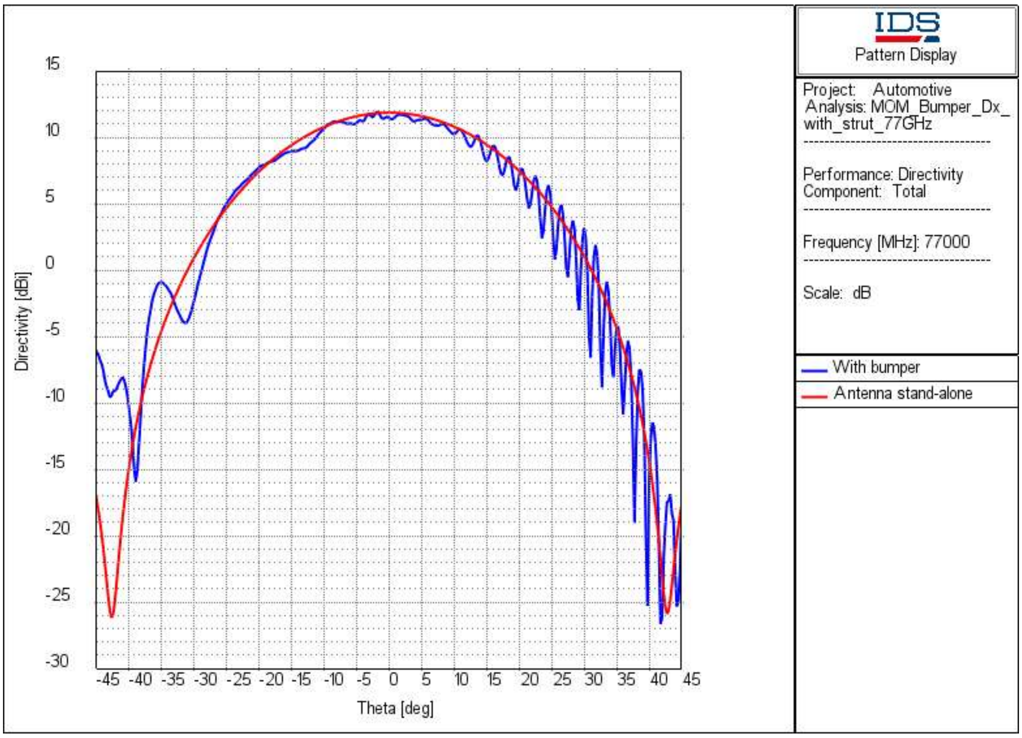


Fig. 4.13 – Free-space vs installed radar antenna pattern

4.4 2.5D MoM Analysis

4.4.1 General Concepts

2.5D MoM Analysis allows the full-wave modelling of planar multilayer (infinite in the XY plane) antenna configurations (2.5D domain).

The radiating elements can be any kind of 3D metallic structures (i.e. metallization embedded in dielectric multi-layers).

Impedance or resistive boundary conditions can be imposed to evaluate the effects of material such as Radar Absorbing materials, thin coated conductors, terrains.

2.5D MoM is a Method of Moments-based electromagnetic code. Two solving techniques are available:

1) **Standard MoM** [BD1]
2) **Sparse Matrix Adaptive Integral Method** (**SM-AIM** [BD8])

The first one solves the standard Method of Moments linear equation system. The second is a special acceleration algorithm that extends the application of the procedure to large structures. Both of them are based on the RWG (Rao-Wilton-Glisson) sub-sectional basis functions representation for the surface parts and on the PWL (Piece-Wise Linear) sub-sectional basis functions representation for the wires (and junction basis functions for modelling their connections). The interaction through the dielectric multi-layers are evaluated using a Green's function formulation for a multi-layered configuration (Mixed-potential integral equation (MPIE) formulation [BD9]). The solution of the problem can be obtained through direct LU factorization for the standard MoM while iterative methods (BiCGStab, GMResR) are required for the SM-AIM.

Among the main features of these techniques are:

- Adoption of a "multi-port" approach to evaluate 'active' observables (i.e. pattern, near-field, coupling) quickly for any kind of excitation configuration;
- Materials that can be managed include: perfectly conductive materials, materials with losses and dielectrics materials;
- R, L, C, Y, Z lumped impedances;
- the geometry of the problem is represented by
  - a "virtual" stack-up consisting of dielectric layers (also with losses) and ideal ground planes ("PEC Layer with aperture") with or without apertures, that we call "Stack Layer"; the mesh is supposed to be immersed in this "virtual" stack-up;
  - the mesh of the CAD model; if the "Aperture" material (4.1.2.3) is used, the mesh represents an "aperture" in an ideal ground plane of the "Stack Layer" (magnetic current representation is used to simulate the aperture EM behaviour);
- special algorithms enable management of multi-scale models (i.e. models in which the ratio between the size of the largest and smallest mesh element is very high) in an effective and accurate way;
- computational cost can be reduced by using the Sparse Matrix Adaptive Integral Method (SM-AIM) [BD8];

The type of material representation managed by the 2.5D MoM code are:

- Perfect Electric Conductor (4.1.2.1)
- Aperture (4.1.2.3)
- Other Physical Materials
    - Multilayer (4.1.2.4)
    - Grid (4.1.2.6)
    - Surface Impedance (4.1.2.7)
    - Tabulated Data (4.1.2.8)

The type of source managed by the 2.5D MoM code is the full-wave port (i.e. the so called ‘Delta Gap’ [BD1]).

2.5D MoM is able to calculate the following output (observables):

- Far Field (4.1.7.5);
- Near Field (4.1.7.6);
- Induced Currents (4.1.7.3);
- S-parameters (4.1.7.1), Coupling (4.1.7.7);
- Input Impedance/Admittance, Return Loss, Reflection Coefficient, VSWR (4.1.7.2);

it can also generate equivalent models (currents based) that can be used as sources in all other available analyses (4.1.7.8).

4.4.2 Application Field

In principle, the full-wave analysis of periodic or aperiodic structures such as arrays, reflect-arrays, transmit-arrays, etc…, can be performed using a standard MoM method.

This approach is highly flexible, i.e. can be applied to any configuration, but at the same time, it is highly demanding in terms of required memory and calculation time.

In practice, the applicability of the standard MoM is limited to small structures.

With the aim to overcome these limitations, we can use the SM-AIM acceleration method.

The main features of these two methods are shortly described below, in order to identify their application fields.

1) **Standard MoM**

   The MoM linear system is usually solved through LU factorization. It is characterized by a high RAM occupancy and high CPU time, therefore, is applicable to the analysis of electrically small structures. It should be noted that solving the system through the LU factorization is particularly efficient for a multiport configuration with a large number of ports since the factorization needs only be computed once. Arbitrary shape 3D metallic objects are also allowed as well as apertures on the ground planes.

2) SM-AIM

SM-AIM is well-suited for the analysis of quasi-planar structures and appears very efficient when applied to multilayer microstrip antennas (e.g. array with periodic or non-periodic lattices).

Being an iterative based method, the issue of convergence is of paramount importance, especially in the presence of complex (multi-scale) geometries that "naturally" lead to high condition numbers. To overcome this difficulty, the Multi-resolution (MR) physics-based pre-conditioner is used to speed up the convergence of the iterative method.

Its memory requirement is very limited and it is, therefore, applicable to the analysis of electrically large structures.

Since the iterative method has to be restarted for each port of the model, if the number of ports is large, the overall time required for this procedure can increase significantly.

Vertical parts in the model being analysed are allowed although it should be considered that the efficiency of this technique is optimal for fully planar structures.

4.4.3 Modelling rules

Modelling rules are those typical of the classical method of moments, that means **the mesh length has to be at most one tenth of wavelength of the medium in which it is embedded**.

To obtain a correct EM representation of the model in terms of basis functions, the geometry should be meshed using a suitable mesh step: for example, the microstrips should be represented by at least three rows of elements; the geometry border should be meshed thicker than the inner part due to the higher current concentration in the edge zone; etc.

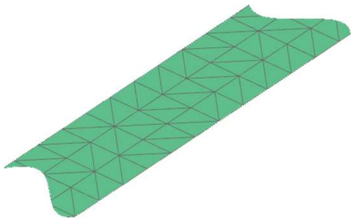


Fig. 4.14 – Mesh example of three rows of meshes in a microstrip

Due to the fact that the 2.5D MoM is mainly used for a planar antenna or EM structure embedded on a dielectric stack-up, the geometrical (i.e. thickness) and EM characteristics of all dielectric materials (multi-layered structure properties) must be provided using the ***"Stack Layer"***. Note that this kind of structure assumes **dielectric layers to be infinite and homogeneous in the XY plane**. This means that dielectric materials are defined by simply specifying their position (relative to upper bound, lower bound and the others layers) and thickness in the layer stack-up.

The mesh and the "Stack Layer" should be defined considering the respective positions between mesh elements and stack-up layers.

Ideal infinite ground planes ("PEC Layer with aperture") can also be defined by assigning their position in the layer stack-up.

Using the "Aperture" material (4.1.2.3) associated to mesh elements, an "aperture" in an ideal ground plane can be simulated; magnetic current representation is used to simulate the aperture EM behaviour.

By default, the layer stack-up contains two layers:

- ***"Lower Bound Layer"***, that describes the characteristic of the lower half space; this material is free space by default.
- ***"Upper Bound Layer",*** that describes the characteristic of the upper half space; this material is free space by default.

Note that the layer stack-up does not need to be modified to simulate a metallic antenna in free space.

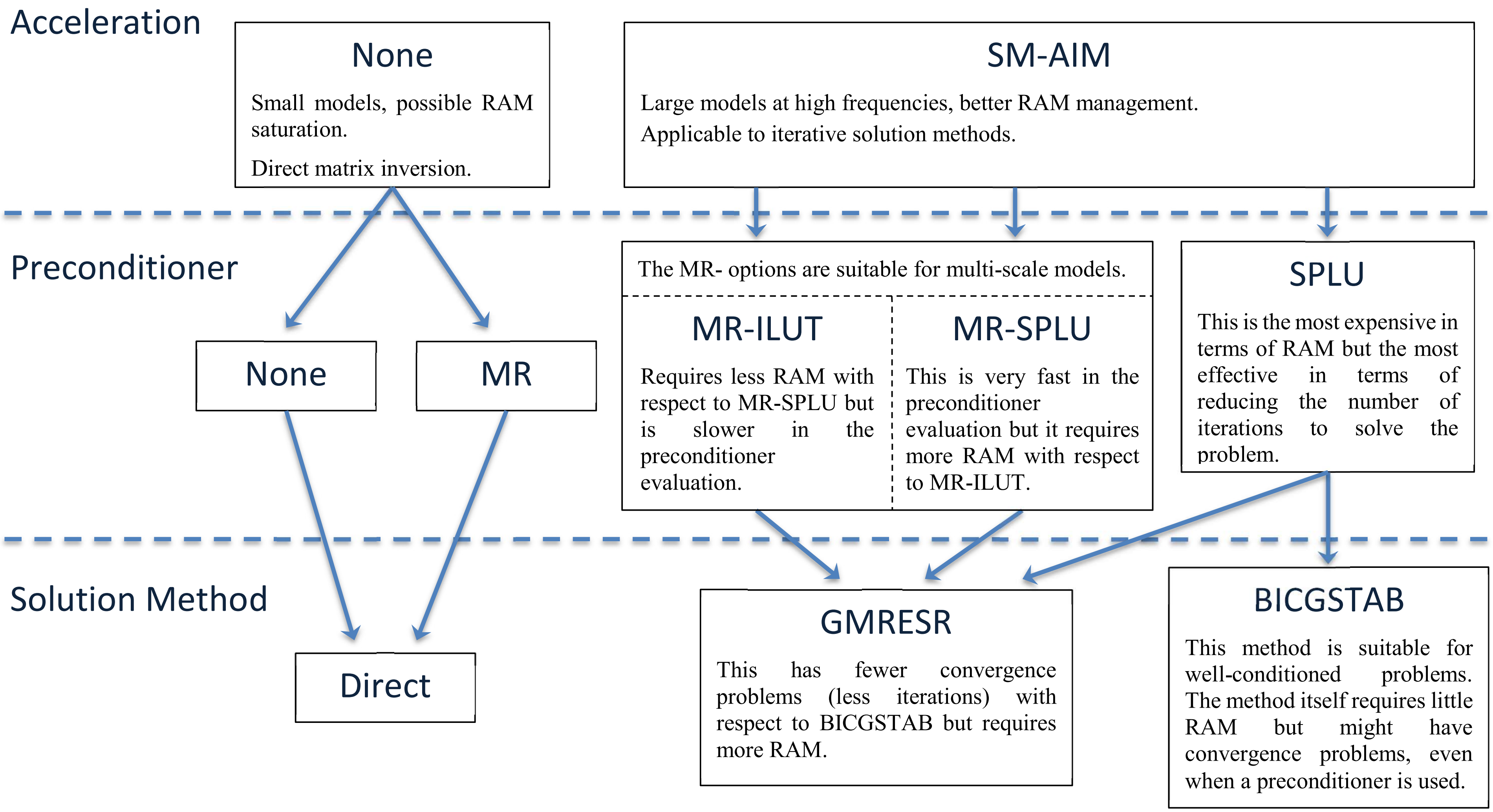


Fig. 4.15 – 2.5D MoM suggested settings

4.4.4 Workflow

Step		Summary
1	Create a FEM and mesh the HF EM model.	Create a FEM using the **Simcenter 3D High Frequency EM Simulation** template. Set the Solver to **Simcenter 3D High Frequency EM** and the **Analysis Type** to **2.5D MoM**. Use the **Stich Edge** function over the whole model. Insert a **Mesh Point** at the end of each wire where there is a connection with a patch. Best results with **Type→Project Point**, **Projection Method→Closest Point**. Create the boundary mesh using the available **2D Triangle** and **1D Wire** elements. Create **High Frequency EM material**. For each mesh collector, choose the appropriate HF EM material and physical properties for your analysis. Create an **EM Field Probe**, if needed. Run the suitable checks, in particular **Duplicate Nodes** and **Duplicate Elements**.
2	Create a simulation	Create a simulation using **Simcenter 3D High Frequency EM** or the **Blank** Simulation template.
3	Create a MoM Solution	Create a new solution with **Solver** set to **Simcenter 3D High Frequency EM** **Analysis Type** set to **2.5D MoM**. **Solution Type** set to **MoM Solution – SYZ-Parameters** Specify solution settings.
4	Create HF EM loads	HF EM loads must be **Full-wave ports**.
5	Create physical properties	Create a **Stack Layer** to model the infinite planar structure at the base of the 2.5D calculation.
6	Create modeling objects	Create a **Frequency scan** to define the solution frequency range. Create a **Geometrical Scan Area** to define where the output must be evaluated, if needed.
7	Create simulation objects	Create a **Lumped Impedance** to add concentrated impedance to the model, if needed.
8	Select solution parameters	Select loads, physical properties, modeling objects and simulation objects for the solution.
9	Solve the solution	Solve the solution to generate the analysis results.
10	Post-process MoM Solution results	Display **S/Y/Z-Parameters** using the HF EM Post-processing tools.
11	Create the solution	Create a new solution with

		Solver set to **Simcenter 3D High Frequency EM** **Analysis Type** set to **2.5D MoM** **Solution Type** set to **Induced Currents**, **Far Field**, **Near Field**, **Coupling**, **Impedance** or **3D Current Distribution**. Select the **MoM Solution** to be used as prerequisite. Create a **Geometrical Scan Area** to define where the output must be evaluated, if needed. Specify solution settings.
12	Select solution parameters	Select loads and simulation objects for the solution.
13	Solve the solution	Solve the solution to generate the analysis results.
14	Post-process results	Display **Induced Currents**, **Far Field**, **Near Field**, **Coupling**, **Impedance** or **3D Current Distribution** using the HF EM Post-processing tools.
15		

Tab. 4-2 – 2.5D MoM workflow

4.4.5 Application examples

An example of a typical application of the Standard MoM and another typical for the SM-AIM are shown in the following.

4.4.5.1 Aperture Coupled Patch Antenna

One of the simplest applications of a 2.5D solver is on aperture coupled patch antennas, which are very common configurations. In this example, a planar radiating element is coupled through a bowtie shaped slot to a microstrip on the bottom of the ground plane. The feeding port is a delta gap assigned between the microstrip and the ground plane. Fig. 4.16 shows the mesh model of this antenna configuration.

The lower dielectric slab has ε_r=3, tanδ=-0.003, thickness=0.51mm. On the upper side there two slabs, with the radiating patch in the middle, the first has ε_r=1.001, tanδ= 0, thickness=10.5mm and the second ε_r=4.35, tanδ= -0.087, thickness=0.95mm.

This is a very small problem (2612 unknowns) that can be easily solved by Standard MoM, e.g. by LU factorization.

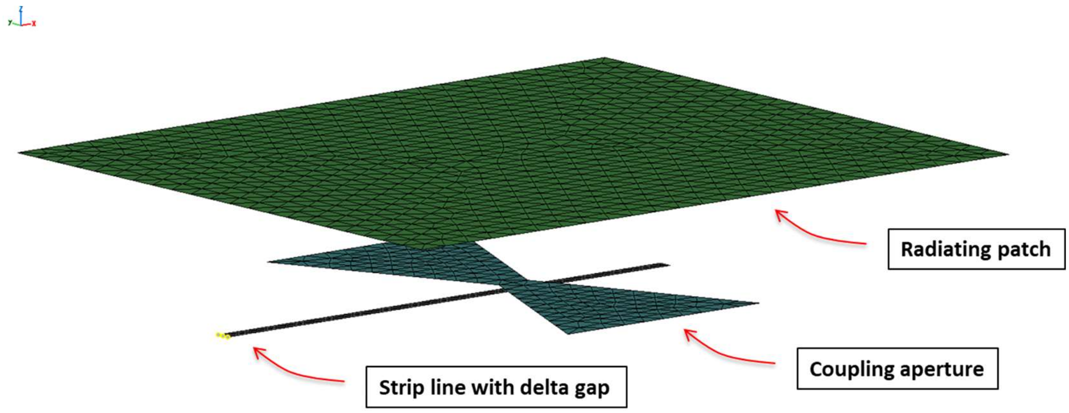


Fig. 4.16 – Aperture coupled patch antenna mesh model

As analysis result, in Fig. 4.17 the reflection coefficient at the port (S11) is calculated in the range 1.3 – 2.5 GHz.

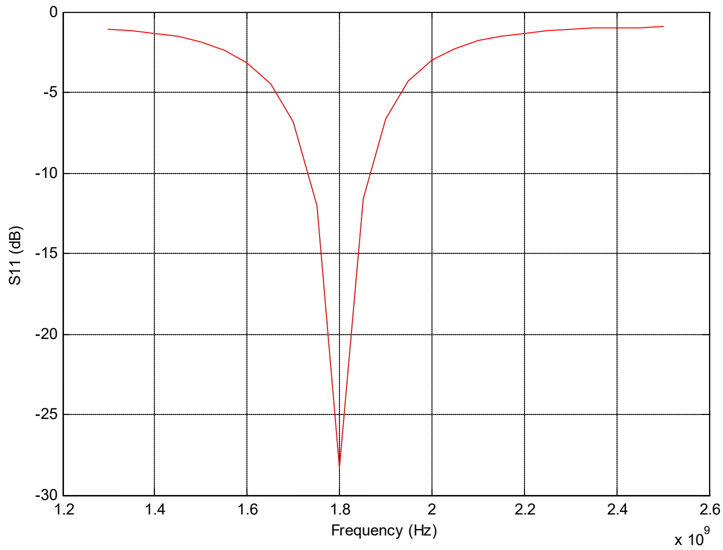


Fig. 4.17 – S11 in the range 1.3 – 2.5 GHz

4.4.5.2 Metasurface Planar Antenna

Metasurface antennas are normally designed by modulating the equivalent reactance of a surface with the aim to convert a bounded surface wave (SW) into an unbounded leaky wave, thus producing a radiated field. One simple way to realize the equivalent reactance is through the use of metallic patches on a dielectric slab. This example is a planar metasurface antenna designed from a modulated anisotropic equivalent reactance radiating a circular polarization in which the technological solution for realizing the metasurface uses dielectric slabs printed with a dense lattice of small metallic patches of variable size and rotation with the aim to create anisotropic boundary conditions. These characteristics mean that the modelling requires a very dense mesh (of the order of lambda/50) and thus generates a very large linear system to be solved, which is typically ill-conditioned. This problem can be managed by the SM-AIM acceleration with a preconditioner.

Fig. 4.18 shows the layout of the antenna consisting of 17593 metallic patches printed on a grounded dielectric slab (ε_r=9.8, tanδ= 0.003, thickness=1.575mm), fed by a surface wave launcher. The patches are slotted circular patches and the launcher is a vertical pin connected to the ground plane (where a delta gap port is assigned) and to a matching element (ringed circular patch), as shown in the figure.

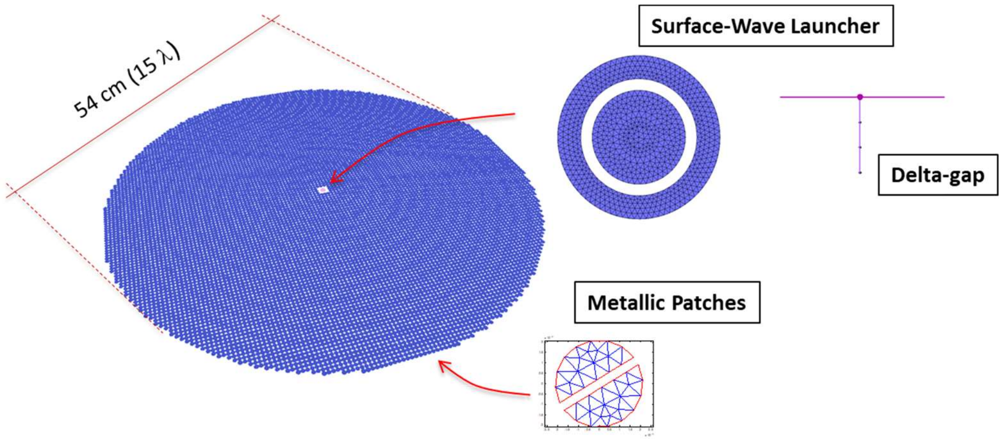


Fig. 4.18 – Layout of the antenna

The problem is of the size of 1233436 unknowns, solved using the SM-AIM method, the BiCGStab iterative solver and with the Multi-Resolution preconditioning algorithm. The solution was reached after 51 iterations and the required memory was about 23GB using the double precision version code. The elapsed time to solve the problem was 1h 13m on an Intel Xeon E5 machine run over 16 cores.

Fig. 4.19 shows the electric currents on the metallic patches, the first image being the spiral distribution required to generate the circular polarization, and the second being the directivity pattern of both right-hand and left-hand components.

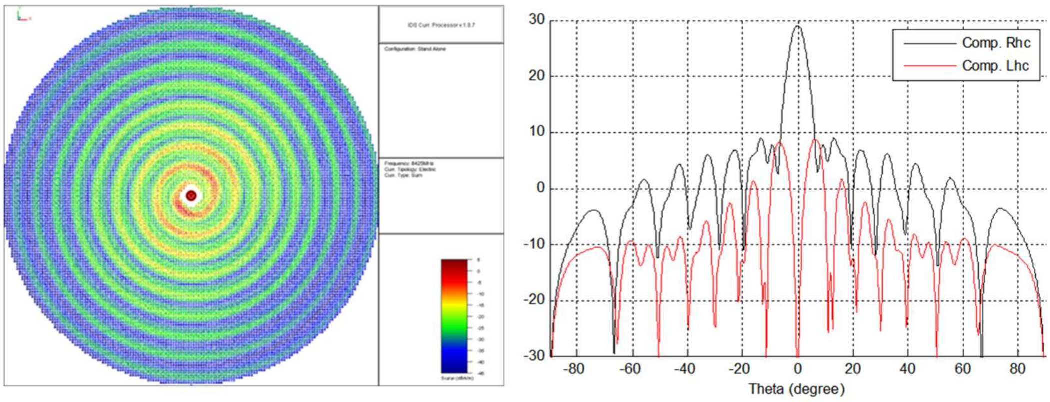


Fig. 4.19 – Electric Currents and Directivity Pattern at 8.425 GHz.

4.5 S-PEEC Analysis

4.5.1 General Concepts

The integral formulations used with the MoM solver are subject to the Low Frequency Breakdown phenomena, leading to a very ill-conditioned linear system. This means that at these frequencies the iterative solvers cannot converge, or a solution with a unacceptable error can be found.

In these circumstances, the S-PEEC method [BD10], [BD11] a surface version of the PEEC method, is used with the aim of mitigating the problem of Low Frequency breakdown.

The PEEC method is an electric field integral equation (EFIE) based approach which can provide an equivalent circuit of the EFIE in terms of the capacitive and inductive interactions between the elemental currents and charges in the discretized structure. The integral equation-based partial, global coupling can be taken one step further, by introducing circuit elements to construct a complete distributed equivalent circuit description of the discrete electromagnetic problem. The resulting circuits are called Partial Element Equivalent Circuits (PEEC). PEEC models have been used extensively to describe discrete approximations of the EFIE in relation to the electromagnetic analysis of interconnections, packaging structures and power systems.

The classical PEEC method relies on an orthogonal discretization of all structures using parallelepipeds as elementary volumes and rectangles as elementary surfaces. This paradigm, which assumes a dominant direction of current flow along the length of conductors, is well suited for interconnect structures. Whenever the scenario involves some arbitrarily shaped structures, the direction of currents is substantially arbitrary. For this reason, a surface version of the PEEC formulation is exploited (S-PEEC), where only the surfaces of conductors are discretized.

The S-PEEC formulation alleviates the LF breakdown problem by keeping separate the unknowns used to expand the surface electric current and charges (this enables separate the magnetic and electric field contributions to the EFIE to be kept separate). The electric currents are expanded on Rao-Wilton-Glisson (RWG) basis functions, and the electric charges are expanded on basis functions that are uniform across the mesh elements.

Unknown surface electric current and charge are related by including the continuity equation in the equation system. The Method of Moment is then applied to obtain a linear system to be solved. For this, **the method shares most of the characteristics of the 3D MoM solver**, with some significant differences that are described in the following.

By alleviating the Low Frequency breakdown problem, the S-PEEC formulation is suitable for Low Frequency applications.

By separately discretising surface electric currents and charges, the S-PEEC formulation results in a higher number of unknowns with respect to the classical MoM on the same mesh. Approximately, for a mesh model having N_e triangles and N_w segments, we have

$$^3/_2\, N_e + N_w - 1 + N_e + N_w = {}^5/_2\, N_e + 2N_w - 1$$

unknowns, instead of $^3/_2\, N_e + N_w - 1$ unknowns of the classical MoM. Due to this reason, in the frequency range where the MoM can be used, S-PEEC will lead to a less efficient solution.

No “volumes” (i.e. dielectric regions) can be defined. The entire structure is presumed to lie in free space.

The MLFMA acceleration scheme is not available for the S-PEEC solver. This is due to the fact that in the frequency range where S-PEEC is usually employed, MLFMA loses its computational efficiency. Adaptive Cross Approximation (ACA) [BD12], [BD13], [BD14] is available to accelerate the matrix-vector product in an iterative solver.

Among the main features:

- a “multi-port” approach is adopted, in order to evaluate ‘active’ observables (i.e. pattern, near-field, coupling) quickly for any kind of excitation configuration;
- the following can be managed: perfectly conductive materials and materials with losses (conductors);
- calculation can be performed with single or double precision variables;
- R, L, C, Y, Z lumped impedances are used;
- an ideal ground plane is assumed;
- the solver is equipped with the Adaptive Cross Approximation (ACA) algorithm to reduce the computational cost;

The S-PEEC code manages conductors. Apart from PEC, you can choose other Physical Material representations provided that they are conductive materials:

- Perfect Electric Conductor (4.1.2.1)
- Other Physical Materials:
    - Multilayer (4.1.2.4)
    - Grid (4.1.2.6)
    - Surface Impedance (4.1.2.7)
    - Tabulated Data (4.1.2.8)

The types of sources managed by the S-PEEC code are:

- Full-wave port (4.1.5.1) (i.e. the so called ‘Delta Gap’ [BD1]);
- Elementary (Electric and magnetic (4.1.5.2) small dipoles, plane waves (4.1.5.3));
- Equivalent models (currents based (4.1.5.6, 4.1.5.7) or SWE (4.1.5.5)).

S-PEEC code is able to calculate the following output (observables):

- Near Field (4.1.7.6);
- Induced Currents (4.1.7.3);
- Electric Potential (4.1.7.4);
- S-parameters (4.1.7.1), Coupling (4.1.7.7);
- Input Impedance/Admittance, Return Loss, Reflection Coefficient, VSWR (4.1.7.2);

it can also generate equivalent models (currents based) that can be used as sources in all other available analyses (4.1.7.8).

4.5.2 Application Field

By alleviating the Low Frequency breakdown problem, the S-PEEC method is usually applied for analysing conducting structure in the frequency range from a few Hz to some MHz.

Typical application includes the analysis of grounding structures, lightning analysis, etc.

4.5.3 Modelling rules

There are no special modelling rules with respect to classical MoM (see 4.3 and 4.4). The application in a low frequency regime may make inadequate the tenth of wavelength rule of thumb. In such cases (a tenth of wavelength being larger than geometrical parts of the structure) an adequate modelling of the structure geometry is mandatory, by considering that discretization is defining the set of basis functions used to expand the equivalent current.

For example, analysing a 10 m long structure (e.g. a fuselage) at the frequency of 1 kHz, we will have a wavelength λ equal to 300 km that is thirty thousand times longer than the structure dimension. To obtain a correct mesh representation of the model a mesh step of about $\lambda/10^6$ or less should be used.

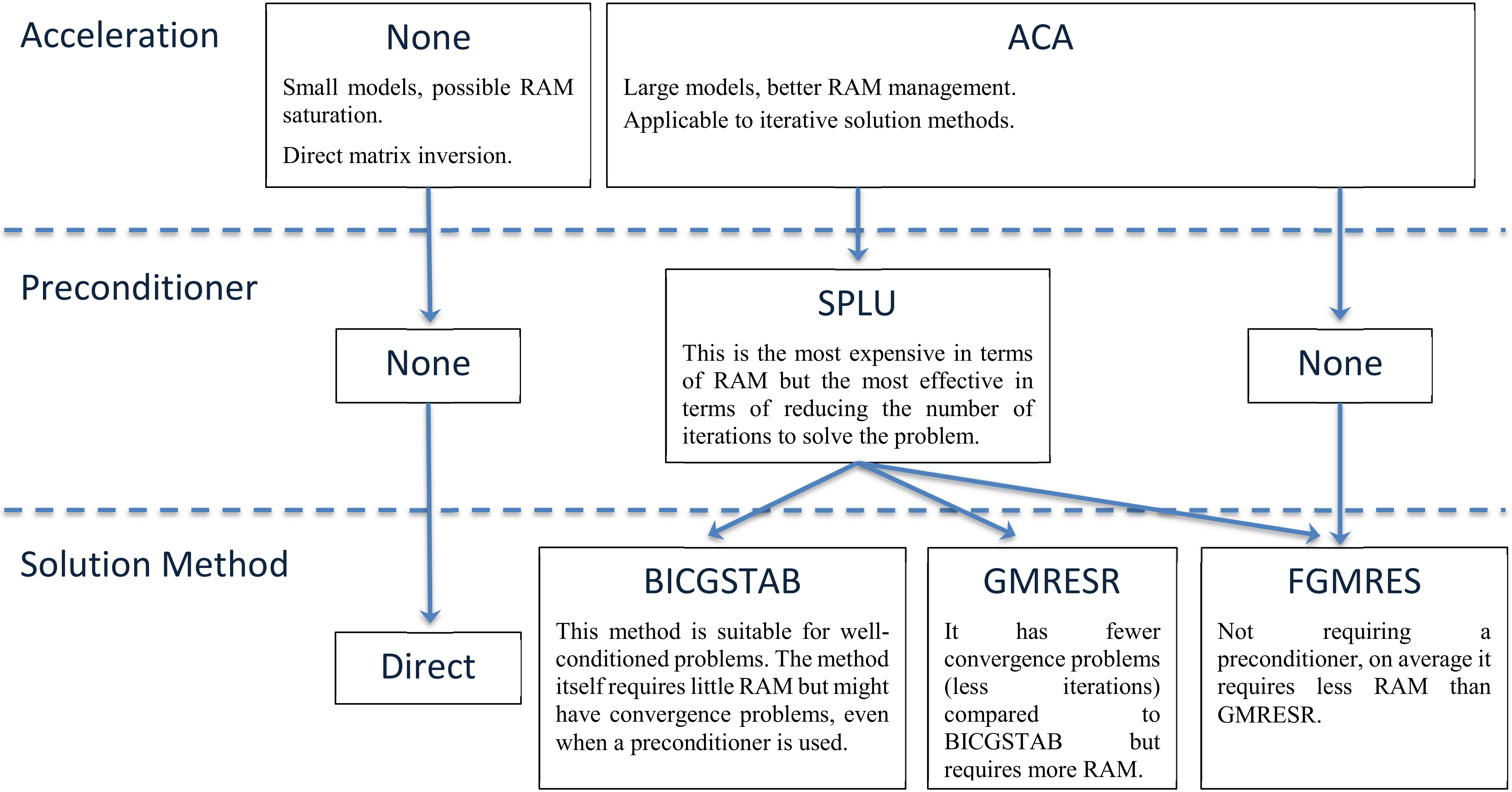


Fig. 4.20 – S-PEEC suggested settings

4.5.4 Workflow

Step		Summary
1	Create a FEM and mesh the HF EM model.	Create a FEM using the **Simcenter 3D High Frequency EM Simulation** template. Set the Solver to **Simcenter 3D High Frequency EM** and the **Analysis Type** to **S-PEEC**. Use the **Stich Edge** function over the whole model. Insert a **Mesh Point** at the end of each wire where there is a connection with a patch. Best results with **Type→Project Point**, **Projection Method→Closest Point**. Create the boundary mesh using the available **2D Triangle** and **1D Wire** elements. Create **High Frequency EM material**. For each mesh collector, choose the appropriate HF EM material and physical properties for your analysis. Create an **EM Field Probe** if needed. Run the suitable checks, in particular **Duplicate Nodes** and **Duplicate Elements**.
2	Create a simulation	Create a simulation using **Simcenter 3D High Frequency EM** or the **Blank** Simulation template.
3	Create a **S-PEEC Solution**	Create a new solution with **Solver** set to **Simcenter 3D High Frequency EM** **Analysis Type** set to **S-PEEC**. **Solution Type** set to **S-PEEC Solution – SYZ-Parameters** Specify solution settings.
4	Create HF EM loads	HF EM loads may be **Full-wave port**, **Electric Dipole**, **Magnetic Dipole**, **EM Plane Wave**, **Synthetic models** such as **Spherical Wave Expansion (SWE)**, **2D Current Distribution** or **3D Current Distribution**.
5	Create modeling objects	Create a **Frequency scan** to define the solution frequency range. Create a **Geometrical Scan Area** to define where the output must be evaluated, if needed.
6	Create simulation objects	Create an **Infinite Plane** to model an ideal ground reference in the XY-Plane at z = 0 if needed. Create a **Lumped Impedance** to add concentrated impedance to the model, if needed.
7	Select solution parameters	Select loads, physical properties, modeling objects and simulation objects for the solution.
8	Solve the solution	Solve the solution to generate the analysis results.

9	Post-process MoM Solution results	Display **S/Y/Z-Parameters** using the HF EM Post-processing tools.
10	Create the solution	Create a new solution with **Solver** set to **Simcenter 3D High Frequency EM** **Analysis Type** set to **S-PEEC**. **Solution Type** set to **Induced Currents**, **Electric Potential**, **Near Field**, **Coupling**, **Impedance** or **3D Current Distribution**. Select the **S-PEEC Solution** to be used as prerequisite. Create a **Geometrical Scan Area** to define where the output must be evaluated, if needed. Specify solution settings.
11	Select solution parameters	Select loads and simulation objects for the solution.
12	Solve the solution	Solve the solution to generate the analysis results.
13	Post-process results	Display **Induced Currents**, **Electric Potential**, **Near Field**, **Coupling**, **Impedance** or **3D Current Distribution** using the HF EM Post-processing tools.
14		

Tab. 4-3 – S-PEEC workflow

4.5.5 Application examples

Two typical examples are considered in the following.

4.5.5.1 Analysis of Current Return Networks

Several situations require the analysis and the verification of grounding structures. As an example, this is the case of an aircraft, where a dedicated conductive electrical pathway, usually named Current Return Network or Almost Equipotential Electrical Network (ALEEN) has to be integrated for the return of direct and alternating currents, faults currents, lightning current, etc. The numerical simulation of ALEEN structures is a very challenging problem. An ALEEN is a large and usually very complicated structure, and it includes small pieces and contacts; we are usually interested in very low frequency (from about a hundred Hertz to some MHz), where classical Method of Moments formulation may become inaccurate due to ill-conditioning problems; capacitive and inductive mutual coupling, skin and proximity effects have to be accurately simulated to precisely estimate impedances and electromagnetic field generated near the structure.

Fig. 4.21 shows pictures of an ALEEN structure (mock-up realized and measured by DEMLAB, Labinal – SAFRAN Group, for testing purposes). Typical analyses include the evaluation of the behaviour of circuits formed by generators, cabling, loads and the ground return. To do this the cables are connected to the grounding structure at the grounding points. Then a port is defined at the generator position and a lumped load at the load position. The S-PEEC solvers, in addition to the electrical quantities evaluated by the canonical MoM solver, can also evaluate the electric scalar potential distribution on the grounding structure and the wires.

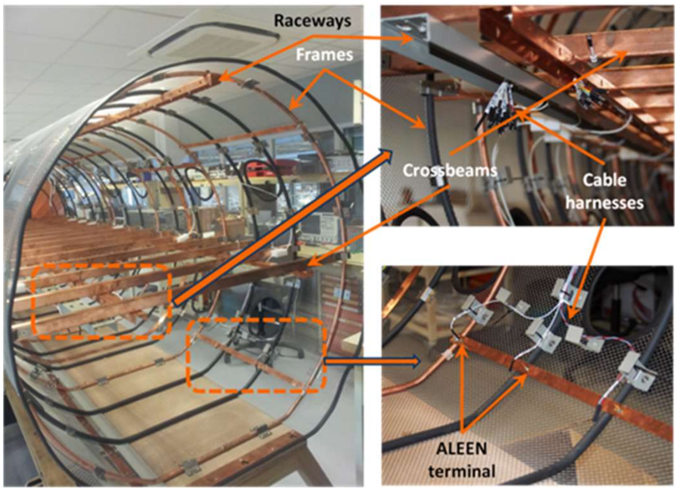


Fig. 4.21 – Example of mock-up structure including an ALEEN

Further analysis include the evaluation of the current flowing on the structure and the electrical potential distribution, when a set of sources and loads are assigned (Fig. 4.22).

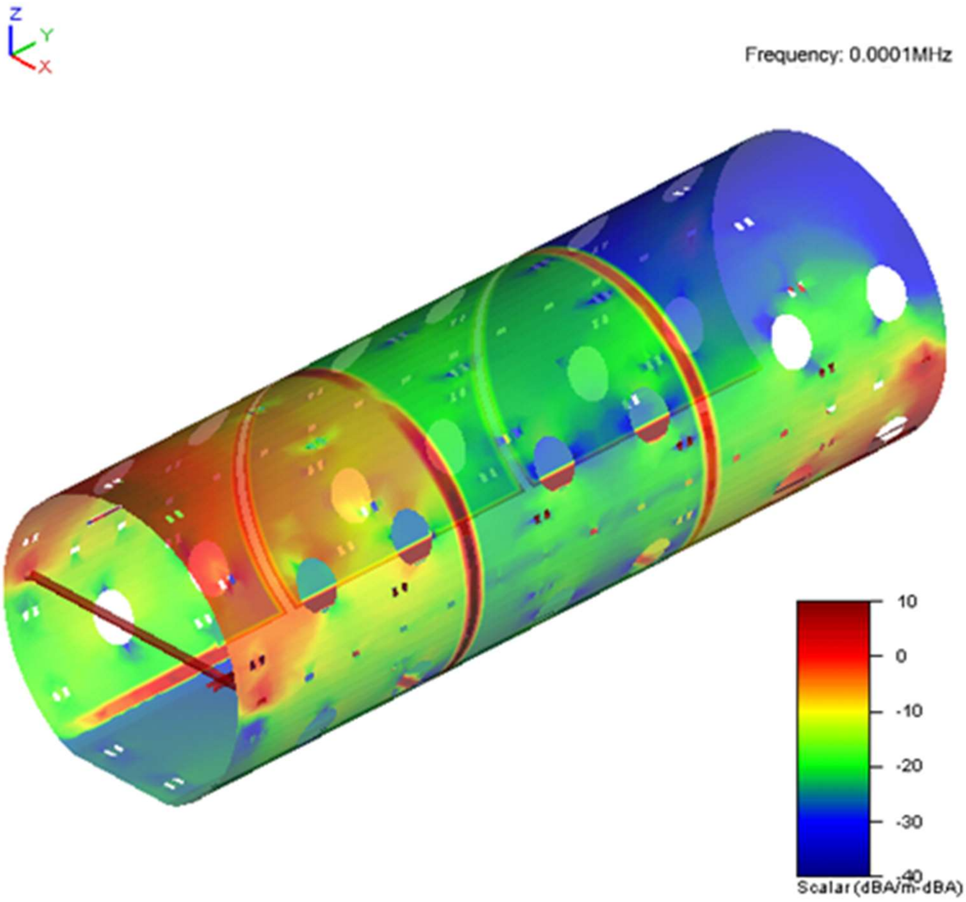


Fig. 4.22 – Example of current flowing on the ALEEN structure

4.5.5.2 Lightning analysis

In this case, the current at a specific position was analyzed, considering this point as the entrance point of the lightning. Then, by defining a spectrum of interest, its effects on loads representing equipment can be estimated in a post-processing phase. Fig. 4.23 shows an example used in validation. In this case, a current is injected on a cable in the EWIS of the mock-up shown in Fig. 4.21, and evaluation is made of the current flowing on another cable.

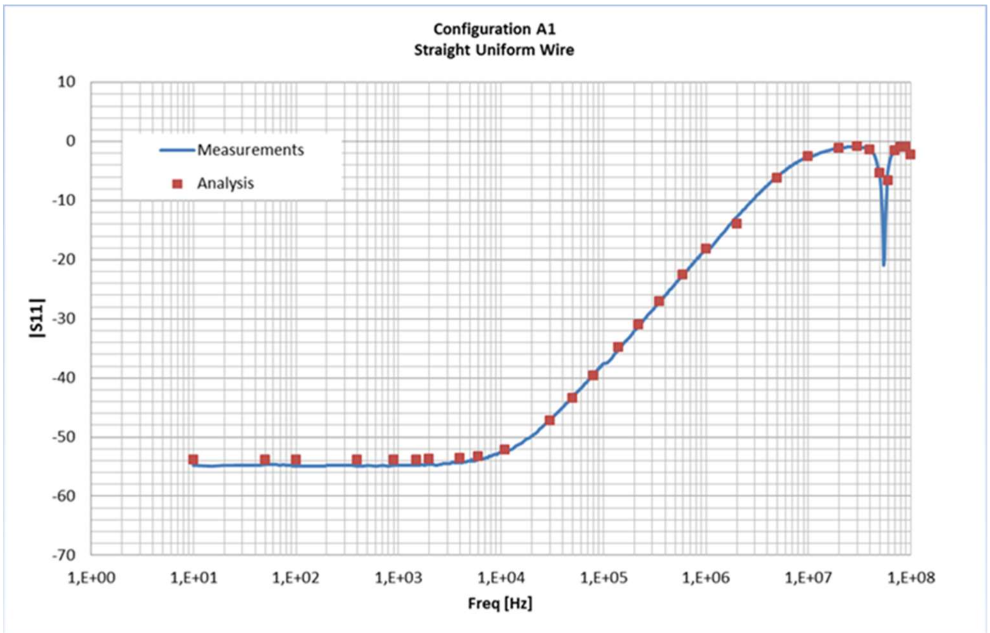


Fig. 4.23 – Measurement vs. simulation for current injection on a cable of the mock-up in Fig. 4.21

4.6 UTD Analysis

4.6.1 General Concepts

The Geometrical Theory of Diffraction (GTD) in its uniform formulation, also known as Uniform Theory of Diffraction (UTD) (see [BD15]) is an electromagnetic theory based on an asymptotic solution of the Maxwell equations.

This method can be applied when evaluating the interaction between a radiating source and a scattering structure whose dimensions are much larger than the field wavelength. In the high frequency range, the scattering obstacle should be at least five wavelengths in size, although reasonably accurate results may be obtained for some bodies even smaller than this. Under these hypothesis, electromagnetic scattering in the microwave frequency range is a “local” phenomenon, much like the scattering of light. The interaction of waves with a body is described as the combination of discrete contributions from a number of “hot points” distributed on the body according to relatively simple geometric laws relating to the propagation of rays, similar to the optics case.

Exact formulations, such as the Method of Moments (MoM) described in section 4.3, are instead required whenever the high-frequency specular, end-region, and diffraction approximations are no longer adequate; that is, when surface traveling, creeping, and edge waves are also important, such as in the resonant region (wavelength comparable to the scattering objects).

The UTD code is based on the Geometrical Theory of Diffraction with extensions for multiple diffraction from straight edges end vertex diffraction.

The primary aspect of the GTD implementation is that its algorithms are separated in two classes:

1) *Ray tracing algorithms* that identify all the paths connecting the source and the observer throughout interactions with the scattering structure. This is essentially a geometrical task that only accounts for the geometrical location of the source and observation points and on the shape and, possibly, material of the scattering objects (the only approximation stands for rays transmitted through a dielectric interface with dielectric constant variable with respect to frequency).

   Each ray starts from the radiating source, hits the scattering body one or more times and then propagates towards the observer location. One particular ray may reach the observer without encountering the body if the body is not placed along the line of sight. Rays must obey the Fermat law of minimum travelling time (in an isotropic medium this coincides with the minimum path length), i.e. for each combination of interactions with the body, the path with minimum length is selected from among the infinite number of paths that are possible.

2) *field computation algorithms* devoted to obtaining the individual ray values and overall values of the electromagnetic field at the observation points. The total scattered field is computed by adding a finite number of contributions associated to the rays by considering:

   - the electromagnetic characteristics of the source;
   - the propagation path length;
   - the scattering process within the structure.

Scattering from the body occurs following several mechanisms associated with the shape of the object and ray impact location. For each path, the power flux is assumed to be concentrated in a flux tube surrounding the *ray* that propagates freely until a scattering structure is encountered. At the interaction point, the wave bounces and a new flux tube emerges from the point. This flux tube will have a different shape to the incoming one, unless the impact point is on a flat surface. Its shape is determined by the local characteristics of the scattered wave, i.e. the wave front curvature and the power flux density. These are in turn a function of those of the incident wave and of the local shape and physical parameters of the object.

Following the locality principle, a ray can only correspond to a small solid angle. The outgoing ray is determined by the direction of the following impact point or of the observation point according to the Fermat law. Therefore, it may correspond to just a small portion of the scattered wave front with a corresponding reduction in power flux density. This is what determines the different scattering mechanisms of GTD.

Each interaction mechanism categorises a different behaviour of the scattered wave, as illustrated by Fig. 4.24 and summarised below:

(a) shows a reflected ray. In this case, if the source is to satisfy the Fermat law, the observer and the reflection point must lie on the same plane. Obviously this geometrical constraint may be impossible to satisfy, i.e. the reflection point would lie outside the object and no reflection would occur.

(b) shows a ray diffracted by a wedge. The scattered ray lies on a cone whose aperture depends on the angle between the incident ray and the wedge axis. Also in this case, this type of contribution may not exist.

(c) shows a ray diffracted by a vertex that exists in the whole portion of space not masked by the object and is therefore always present.

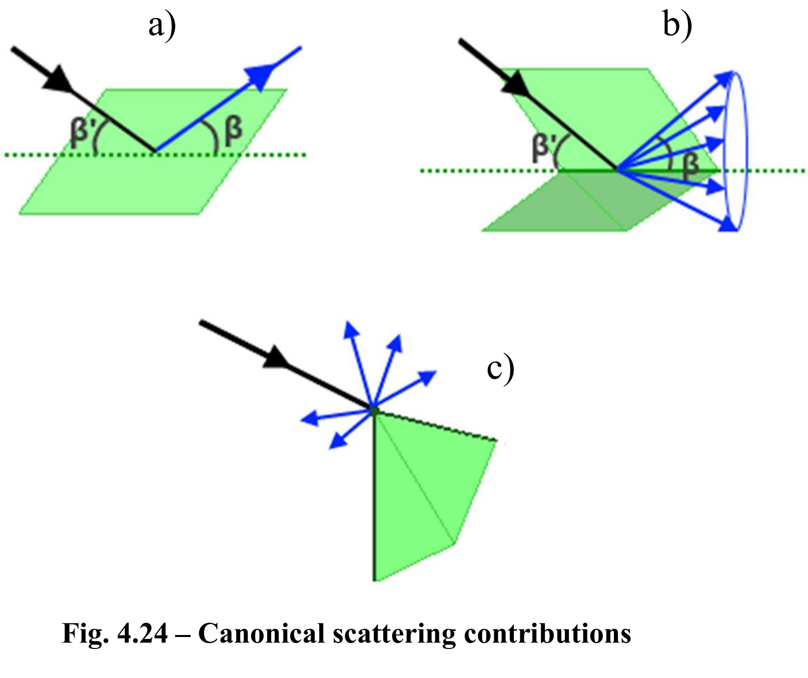


Fig. 4.24 – Canonical scattering contributions

The scattered field associated with each scattered ray is computed by multiplying the incident field by a canonical coefficient that depends, as indicated above, on

- interaction mechanism (reflection, diffraction, etc.);
- local characteristics of the geometry, relative direction of incidence and scattering.

The magnitude of the coefficients is a function of fractional powers of the free space propagation constant (wave-number), giving rise to a classification of the different contribution according to their order. The higher the order of the contributions included, the more accurate the field estimation.

GTD algorithms are founded on well-known formulations and on the most recent definitions of these coefficients, based on the state-of-the-art electromagnetic formulations obtained from co-operation with the research group of Profs. R. Tiberio and S. Maci of the Universities of Siena and Florence ([BD22], [BD23], [BD24]).

Among these canonical contributions we can note:

- reflected by plate surface ([BD15], [BD16]);
- diffraction by wedge (edge) ([BD15], [BD16], [BD17]);
- diffraction by vertex (from [BD18] to [BD23]);

Among the main features:

- the following materials can be managed: perfectly conductive materials, materials with losses and dielectrics materials;
- ideal ground plane is used (edges of the mesh elements corresponding to the ground should be placed above the ground itself to avoid edge contributions);
- special algorithms permit acceleration of the source radiation to avoid high computational cost with complex loads.

This method is provided with extensions to support dielectric, stratified materials and rough surfaces. The materials that the formulation can consider through their transmission and reflection coefficients, are:

- Perfect Conductor (4.1.2.1)
- Other Physical Materials
    - Half Space (4.1.2.5)
    - Multilayer (4.1.2.4)
    - Grid (4.1.2.6)
    - Surface Impedance (4.1.2.7)
    - Tabulated Data (4.1.2.8)

The types of sources managed by the UTD code are:

- Pattern (4.1.5.4);
- Elementary (Electric and magnetic (4.1.5.2) small dipoles);
- Equivalent models (currents based 4.1.5.6, 4.1.5.7 or SWE (4.1.5.5)).

UTD code is able to calculate the following output (observables):

- Far Field (4.1.7.5);
- Near Field (4.1.7.6);
- Coupling (4.1.7.7) (using Field-Current or Friis formulation);

it can also generate equivalent models (currents based from Near Field data) that can be used as sources in all other available analyses (4.1.7.8).

4.6.2 Application Field

The UTD method is applicable in the evaluation of the interaction between a radiating source and a scattering structure whose dimensions are much larger than the field wavelength. In the high frequency range, the scattering obstacles should be at least five wavelengths in size, although reasonably accurate results may be obtained for some bodies even smaller than this.

Typical application includes the analyse of large structures in terms of wavelength, such as ships, vehicles or scenario configurations, like airports, factories, cities, etc.

4.6.3 Modelling Rules

The first step of the modelling procedure is the mesh creation starting from a "Platform CAD model". The meshed model or "electromagnetic model" must consist of elementary facets (quadrilaterals and triangles), whose dimensions should be as large as possible but small enough to suitably fit the surface curvature of the platform geometry.

A unit normal vector pointing outward, the scatterer associated to each mesh elementary facet and a check on the normal pointing direction must all be checked and completed before saving the meshed model. Note that only faces with outward pointing normals participate in the scattering phenomenon.

When creating the electromagnetic model, some further checks are necessary to avoid disconnected plates due to modelling errors, which are due to an incorrect definition of the ideally coincident vertexes and edges of the plates. Often this is due to errors in the vertex co-ordinates.

A tricky case is that of two plates, which should form a wedge that have opposite normal as shown in Fig. 4.25. The UTD solver interprets the wedge as two coincident edges. This produces two major drawbacks:

- a ray could propagate through the wedge;
- the canonical coefficient to evaluate the scattering field is that of an edge and, in general, it has a larger amplitude than that of the wedge.

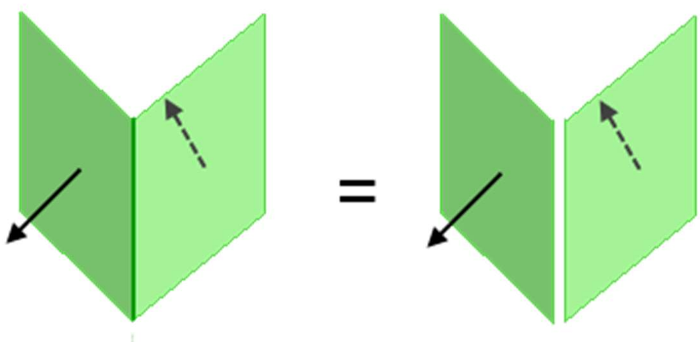


Fig. 4.25 – Disconnected plates because of opposite normal condition

Another tricky case is that of two plates, which form a T-structure as shown in Fig. 4.26. The UTD solver interprets the two internal wedges as a single edge. Also in this case, as a major drawback, a ray could propagates through the wedge.

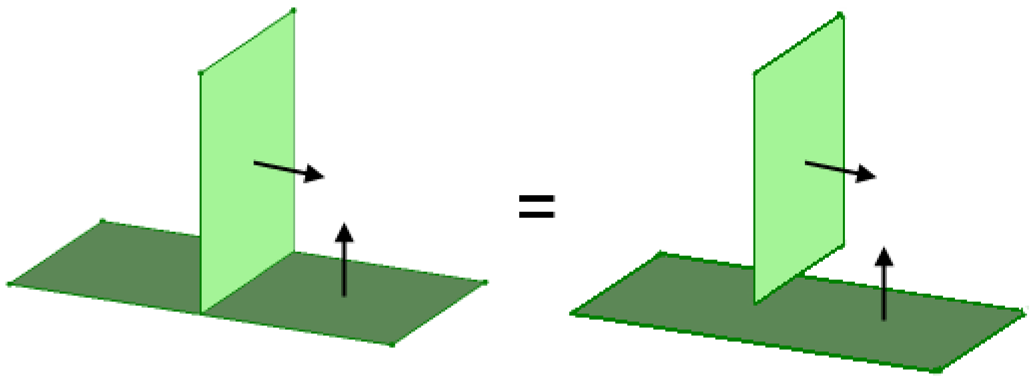


Fig. 4.26 – Disconnected plates arising from a T-structure with their normals

This case typically occurs when a plate is placed on an ideal ground. In order to avoid the appearance of unphysical disconnected edges, the system prevents a side of a polygon from leaning on the ground and requires the polygon to be intersected with it.

The second step of the modelling procedure is antennas installation, and each time we take this step we have to answer the following questions:

1) what is the platform dimension with respect to the wavelength (i.e. the frequencies of analysis)?
2) where is my antenna installed (i.e. does my antenna verify the far field condition with respect to the surrounding structures)?

Depending on the platform dimension with respect to the wavelength and on the distance between the antenna and the closest scattering structure, the correct modelling procedure has to be selected and the most appropriate antenna model has to be used.

The UTD modelling procedure can be used if the interaction between the platform and the antenna current distribution is weak. This approximation can be well suited at high frequency regimes (the antenna –platform interaction decreases significantly because the scattering becomes more a localized phenomenon than a collective phenomenon). The simulation is performed using an "equivalent model" of the antenna and evaluating the field scattered by the mesh of the platform: the "total" field is the superposition of the UTD scattering contributions.

The UTD method can handle different "equivalent models" source to be used depending on the modelling conditions:

- **Pattern** (4.1.5.4): this point source model is well suited when the scattering structures and the observation point are located in the far field region (Fraunhofer's region);
- **Spherical wave expansion** (4.1.5.5): this point source model is well suited to calculate near and far field outside of the smallest sphere between the one surrounding the source (i.e. the physical encumbrance of the antenna) and the one of characteristic radius (*r*) of the SWE itself.
- **2D current distribution** (4.1.5.6): this is an extended source model represented by a collection of elementary electric and magnetic sources defined on a rectangular aperture. From this model, both near field and far field may be determined accurately.

- **3D current distribution** (4.1.5.7): this is an extended source model represented by a collection of elementary electric and magnetic sources defined on a triangular mesh of an arbitrary surface in 3D space. From this model, both near field and far field may be determined accurately.
- **Small dipoles (electric or magnetic)** (4.1.5.2): this point source model, represented by small ideal dipoles, is a 1D current distribution. From this model both near field and far field may be determined accurately.

Note that only far fields are used for GTD calculations because a “local plane wave approximation” is assumed and the field components are transverse to the direction of propagation.

For current distribution “equivalent models” sampled at Nyquist rate, each elementary source has a maximum size shorter than half of the wavelength of the radiation they emit (i.e., "electromagnetically short" antennas) hence the far region boundaries are measured in terms of a simple ratio of the distance r from the radiating source to the wavelength λ of the radiation. For such an antenna, the far-field is the region for which $r \gg 2\lambda$. A constraint on the minimum distance d_{min} between the platform (or the observer) and the nearest elementary source: $d_{min} \gg 2\lambda$ can be derived from this relationship. Empirically, reasonably accurate results can be obtained if $d_{min} > 5\lambda$ and even at shorter distances in the hypothesis that the impinging field on the nearest mesh elements is negligible with respect to the field level in the main radiating direction.

4.6.4 Workflow

Step		Summary
1	Create a FEM and mesh the HFEM model.	Create a FEM using the **Simcenter 3D High Frequency EM Simulation** template. Set the Solver to **Simcenter 3D High Frequency EM** and the **Analysis Type** to **UTD**. Create the boundary mesh using the available **2D Triangle** and **2D Quad** elements. Create **High Frequency EM material**. For each mesh collector, choose the appropriate HF EM material and physical properties for your analysis. Create an **EM Field Probe**, if needed. Run the suitable checks, in particular **2D Element Normals**, **Duplicate Nodes** and **Duplicate Elements**.
2	Create a simulation	Create a simulation using **Simcenter 3D High Frequency EM** or the **Blank** Simulation template.
3	Create the solution	**Solver** set to **Simcenter 3D High Frequency EM** **Analysis Type** set to **UTD** **Solution Type** set to **Far Field**, **Near Field**, **Coupling** or **3D Current Distribution** Specify solution settings.
4	Create HF EM loads	HF EM loads may be **Electric Dipole**, **Magnetic Dipole** or **Synthetic models** such as **Pattern**, **Spherical Wave**

		Expansion (SWE), **2D Current Distribution** or **3D Current Distribution**.
5	Create modeling objects	Create a **Frequency scan** to define the solution frequency range. Create a **Geometrical Scan Area** to define where the output must be evaluated, if needed.
6	Create simulation objects	Create an **Infinite Plane** to model an ideal ground reference in the XY-Plane at z = 0, if needed.
7	Select solution parameters	Select loads, modeling objects and simulation objects for the solution.
8	Solve the solution	Solve the solution to generate the analysis results.
9	Post-process results	Use post-processing tools, such as contour plots or XY graphing to analyse your results. Use HF EM Post-processing tools to display your output. Display **Far Field**, **Near Field**, **Coupling**, **Impedance** or **3D Current Distribution**.

Tab. 4-4 – UTD workflow

4.7 IPO Analysis

4.7.1 General Concepts

Iterative Physical Optics (IPO) is an iterative high-frequency technique for evaluating the interaction between a radiating source and a scattering structure whose dimensions are larger than the field wavelength and is used because it avoids the incorrect behaviour of others asymptotic techniques (e.g. UTD) near caustics and boundaries.

It was originally developed for analyzing the scattering from open-ended cavities with perfectly electrically conducting (PEC) walls [BD26], [BD27]. In particular, it was developed to analyze arbitrarily shaped cavities for which analytical waveguide modal methods [BD28] are not applicable. It was then extended in [BD29]-[BD31] to the case of impedance boundary conditions, and [BD32] investigated the possibility of analyzing dielectric thin slabs. Moreover it was applied to compute the scattered field and the radar cross section of electrically large and realistic complex targets [BD33], [BD34], such as tanks, airplanes, etc.

With the aim of electrically analyzing ever larger objects, other upgrades have been made to reduce the computational burden and to accelerate convergence of the IPO algorithm. Techniques based on the domain decomposition of the scatterer surface were introduced in [BD30], [BD36], [BD38]. In particular, in [BD30] and in [BD36] the Fast Far-Field Approximation (FaFFA) algorithm [BD39], [BD41], [BD42] was adopted for accelerating the computational burden relevant to the computation of the interactions between the various elements in the scenario under analysis at each iteration. Furthermore, in [BD30], [BD36], iterative relaxation techniques, such as the Jacobi Minimal Residual (JMRES) [BD35], were used in order to control the convergence of the IPO algorithm.

The NVIDIA CUDA compute platform has also been used [BD39] to push forward the state of the art of IPO code and to further enhance its computational capabilities through use of Graphics Processing Units (GPUs).

The Iterative Physical Optics (IPO) algorithm is based on the application of the equivalence theorem for the description of the scattering of a complex scenario. The **equivalent currents** are estimated by using the Physical Optics (PO) approximation for both impenetrable (PEC or impedance boundary condition) and penetrable (multilayered electrically thin slabs) objects.

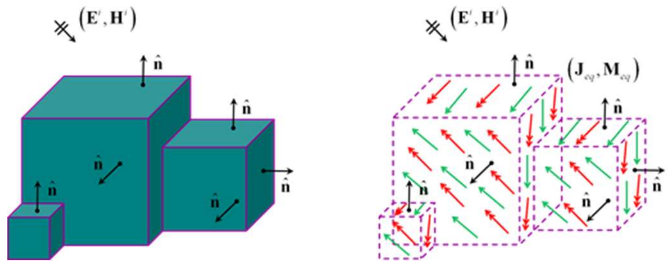


Fig. 4 –Equivalence theorem for IPO algorithm

The iterative process allows a reconstruction of the interactions between the objects without resorting to ray-tracing. In particular, the algorithm reconstructs the reflections from objects and the forward scattering which produces a shadow behind an object, and then the masking of the incident field on another object or portion of it located behind the first. At each step, further reflection/masking is introduced to the description of scattering. The estimate of equivalent currents is therefore similar to that produced by a tracking algorithm in Geometrical Optics (GO) rays up to an order of interaction equal to the number of steps in the IPO algorithm. However, the IPO algorithm avoids the ray tracking operation which is replaced by the calculation of the scattered field from the surfaces at each iteration. In addition, the IPO algorithm, compared to the multiple-reflection GO ray-based algorithm, also introduces diffractive contributions (in PO approximation) which, although not asymptotically correct, avoid the sharp boundaries which are present in the estimate of GO ray-based current.

Among the main features:

- the following materials can be managed: perfectly conductive materials, materials with losses and dielectrics materials;
- ideal ground plane is used;
- the EM behaviour of the structure is stored in the Induced Currents solution; all the other solutions use the Currents as a pre-requisite to be solved and the results are given as superimposition of the Currents effects and the direct incident source contribution;
- special algorithms enable acceleration of the source radiation to avoid high computational costs with complex loads;
- the solver is equipped with the FaFFA algorithm and GPU acceleration to reduce the computational costs;

The type of material representations managed by the IPO code are:

- Perfect Conductor (4.1.2.1)
- Other Physical Materials
    - Half Space (4.1.2.5)
    - Multilayer (4.1.2.4)
    - Grid (4.1.2.6)
    - Surface Impedance (4.1.2.7)
    - Tabulated Data (4.1.2.8)

The types of sources managed by the IPO code are:

- Pattern (4.1.5.4);
- Elementary (electric and magnetic (4.1.5.2) small dipoles, plane waves (4.1.5.3));
- Equivalent models (currents based 4.1.5.6, 4.1.5.7 or SWE (4.1.5.5)).

IPO code can calculate the following output (observables):

- Induced Currents (4.1.7.3): this is the IPO Solution used to calculate the other outputs (pre-requisite for the other solutions);
- Far Field (4.1.7.5);
- Near Field (4.1.7.6);
- Coupling (4.1.7.7) (using Field-Current or Friis formulation);

it can also generate equivalent models (currents based from Near Field data) that can be used as sources in all other available analyses (4.1.7.8).

4.7.2 Modelling Rules

The first step of the modelling procedure is mesh creation starting from a "Platform CAD model". The meshed model or "electromagnetic model" must consist of elementary facets (quadrilaterals and triangles), whose dimensions should be as large as possible but small enough to suitably fit both the surface curvature and the amplitude and phase variations of the incident field. Therefore the mesh step suggested for the IPO method is about **16 facets per square wavelength**, that means that the **mesh length has to be at most a quarter of wavelength** of the medium constituting the structure.

A unit normal vector pointing outward, the scatterer associated to each mesh elementary facet, and a check on the normal pointing direction must all be checked and completed before saving the meshed model. Note that only faces with the outward pointing normals participate in the scattering phenomenon.

In the presence of an ideal ground, remember that only the part of the electromagnetic model lying above it will be considered in the simulation. Moreover, a polygon or a side of it, can lie on the ground plane.

The second step of the modelling procedure is antennas installation, and each time we take this step we have to answer the following questions:

1) what are the platform dimensions with respect to the wavelength (i.e. the frequencies of analysis) ?
2) where is my antenna installed (i.e. does my antenna verify the far field condition with respect to the surrounding structures) ?

Depending on the platform dimensions with respect to the wavelength and on the distance between the antenna and the closest scattering structure, a correct modelling procedure has to be selected and the most appropriate antenna model has to be used.

The IPO modelling procedure can be used if the interaction between the platform and the antenna current distribution is weak. This approximation can be well suited at high frequency regimes (the antenna –platform interaction decreases significantly because the scattering becomes more a localized phenomenon than a collective phenomenon). The simulation is performed using an "equivalent model" of the antenna and evaluating the field scattered by the mesh of the platform: the "total" field is the superposition of the incident and scattered contributions.

The IPO method can handle different "equivalent model" sources depending on the modelling conditions:

- **Pattern**: this point source model is well suited when the scattering structures and the observation point are located in the far field region (Fraunhofer's region);
- **Spherical wave expansion** (4.1.5.5): this point source model is well suited to calculate near and far field outside of the smallest sphere between the one surrounding the source (i.e. the physical encumbrance of the antenna) and the one of characteristic radius (*r*) of the SWE itself.
- **2D current distribution** (4.1.5.6): this is an extended source model represented by a collection of elementary electric and magnetic sources defined on a rectangular aperture. This model can accurately determine both near field and far field .

- **3D current distribution** (4.1.5.7): this is an extended source model represented by a collection of elementary electric and magnetic sources defined on a triangular mesh of an arbitrary surface in 3D space. This model can accurately determine both near field and far field.
- **Small dipoles (electric or magnetic)** (4.1.5.2): this point source model, represented by small ideal dipoles, is a 1D current distribution. This model can accurately determine both near field and far field.
- **Plane waves** (4.1.5.3): the location of the source doesn't matter, it is only used for visualization purposes; the direction of the wave is defined by a vector, the wave is an infinite plane wave-front orthogonal to the coming direction.

Note that PO structure currents are determined from the radiative near-field expressions, including the r-component, of the electric and magnetic fields.

For current distributions sampled at Nyquist rate, each elementary source has a maximum size shorter than half of the wavelength of the radiation they emit (i.e., "electromagnetically short" antennas) hence the far and near region boundaries are measured in terms of a simple ratio of the distance r from the radiating source to the wavelength λ of the radiation. For such an antenna, the radiative near-field is the region for which $r > \lambda/2\pi$ and $r < \lambda$ and the far-field is the region for which $r \gg 2\lambda$. From these relationships, we can derive a constraint on the minimum distance d_{min} between the platform (or the observer) and the nearest elementary source: $d_{min} > \lambda/2\pi$.

The above condition only applies however to perfectly conductive structures.

If dielectric materials are present, the reflection and transmission Fresnel coefficients can only be accurately computed in the hypothesis of incident "locally plane wave". With this approximation, the far-field condition must be verified and the constraint on the minimum distance d_{min} , between the platform (or the observer) and the nearest elementary source, becomes more stringent: $d_{min} \gg 2\lambda$. Empirically, reasonably accurate results can be obtained if $d_{min} > 5\lambda$, and even at shorter distances, in the hypothesis that the impinging field on the nearest mesh elements is negligible with respect to the field level in the main radiating direction.

4.7.3 Application Fields

The IPO is an iterative high-frequency technique applicable when evaluating the interaction between a radiating source and a scattering structure whose dimensions are larger than the field wavelength and is used to avoid the incorrect behaviour seen with other asymptotic techniques near caustics and boundaries.

Typical applications include the analyse of structures larger than one wavelength, such as antenna reflectors, antenna co-siting, radomes, vehicles, etc.

4.7.4 Workflow

Step		Summary
1	Create a FEM and mesh the HF EM model.	Create a FEM using the **Simcenter 3D High Frequency EM Simulation** template. Set the Solver to **Simcenter 3D High Frequency EM** and the **Analysis Type** to **Iterative PO**. Use the **Stich Edge** function over the whole model.

		Create the boundary mesh using the available **2D Triangle** and **2D Quad** elements. Create **High Frequency EM material**. For each mesh collector, choose the appropriate HF EM material and physical properties for your analysis. Create an **EM Field Probe**, if needed. Run the suitable checks, in particular **2D Element Normals**, **Duplicate Nodes** and **Duplicate Elements**.
2	Create a simulation	Create a simulation using **Simcenter 3D High Frequency EM** Simulation template.
3	Create an **IPO Solution**	Create a new solution with **Solver** set to **Simcenter 3D High Frequency EM** **Analysis Type** set to **Iterative PO** **Solution Type** set to **Induced Currents (IPO Solution)** Specify solution settings.
4	Create HF EM loads	HF EM loads may be **Electric Dipole**, **Magnetic Dipole**, **EM Plane Wave**, **Synthetic models** such as **Pattern, Spherical Wave Expansion (SWE)**, **2D Current Distribution** or **3D Current Distribution**.
5	Create modeling objects	Create a **Frequency scan** to define the solution frequency range. Create a **Geometrical Scan Area** to define where the output must be evaluated, if needed.
6	Create simulation objects	Create an **Infinite Plane** to model an ideal ground reference in the XY-Plane at z = 0, if needed.
7	Select solution parameters	Select loads, physical properties, modeling objects and simulation objects for the solution.
8	Solve the solution	Solve the solution to generate the analysis results.
9	Post-process MoM Solution results	Display **Induced Currents** using the HF EM Post-processing tools.
10	Create the solution	Create a new solution with **Solver** set to **Simcenter 3D High Frequency EM** **Analysis Type** set to **Iterative PO** **Solution Type** set to **Far Field**, **Near Field**, **Coupling** or **3D Current Distribution**. Select the **Induced Currents (IPO Solution)** to be used as prerequisite. Create a **Geometrical Scan Area** to define where the output must be evaluated, if needed. Specify solution settings.

11	Select solution parameters	Select loads and simulation objects for the solution.
12	Solve the solution	Solve the solution to generate the analysis results.
13	Post-process results	Display **Far Field**, **Near Field**, **Coupling**, **Impedance** or **3D Current Distribution** using the HF EM Post-processing tools.

Tab. 4-5 – IPO workflow

4.7.5 Application examples

This section provides some examples of the IPO solver to demonstrate how it can be applied to various kinds of problems of interest to the EM community. This issue is addressed by also investigating application fields other than those that the IPO was originally designed to address. Advantages and limitations of the method are therefore discussed in order to define IPO within in the scenario of computational EM for antennas and platforms.

The following scenarios will be considered in the following subsections:

- the EM analysis of reflector antennas in different applications;
- the EM analysis of radomes;
- problems of antenna placement.

All these results will show the effectiveness and the capabilities of the developed tool.

4.7.5.1 Reflector Antennas

The PO algorithm is usually applied to model reflector antennas, with sequential multi-bounce approach for multi-reflectors antennas. While being conceptually simple, such approach requires user expertise in the selection of the multi-bounce path. On the contrary, IPO automatically analyses all possible bouncing paths, stopping the computation on the basis of a convergence threshold, while maintaining the same computational complexity. Apart from this simplification, IPO is therefore particularly useful for managing those non-standard configurations in which the interactions among different parts of the antenna are too complex to be “pre-selected” by the user.

For large reflector systems, the proposed IPO algorithm was first used to simulate a 30 m Near Field Cassegrain antenna up to 10 GHz with a mixed (full-wave/IPO) method approach. The huge dimension of the problem – the main reflector diameter is 103 λ at 10 GHz – requires highly efficient parallel implementation of the algorithms. A detailed description of the procedure used in the simulations along with the main results can be found in [BD42].

Two further other reflector systems are analysed in the following.

4.7.5.1.1 Single Reflector with Struts

This section describes the analysis of a single reflector with a circularly symmetric parabolic surface shape fed by a horn. The feed horn is held in place at the focus by three struts. The example shows the significance of including the scattering from these supports in the analysis. The antenna is designed to operate in the Ka band at 30 GHz and has the following geometrical characteristics: diameter D = 500 mm; focal length F = 250 mm; subtended angle from the focal point θ = 53.1°; strut diameter d = 10 mm.

The two main effects of the struts scattering are the following:

- The feed field is blocked by the struts which create a shadow on the main reflector in the region between the struts and the reflector rim (spherical wave blockage) (inset in Fig. 4.27). This effect reduces the aperture efficiency and consequently the peak directivity. Furthermore, the field is scattered onto the antenna where it will be reflected in a direction away from boresight. This usually gives rise to an increase in the near-in side-lobe level as we can see in Fig. 4.28 showing a zoom in the near main beam angular region.
- The struts also block the field reflected from the dish, resulting also in a loss of the on-axis directivity (plane wave blockage). The strut scattered field causes a side-lobe increase along the so-called "Keller's cone", a cone with axis along the strut and an opening angle defined by the angle between the strut and the reflector boresight axis. In the present geometry, the main strut scattering direction is 48° with respect to the boresight axis. As a matter of the fact, an increase in the side-lobe level is shown in Fig. 4.27.

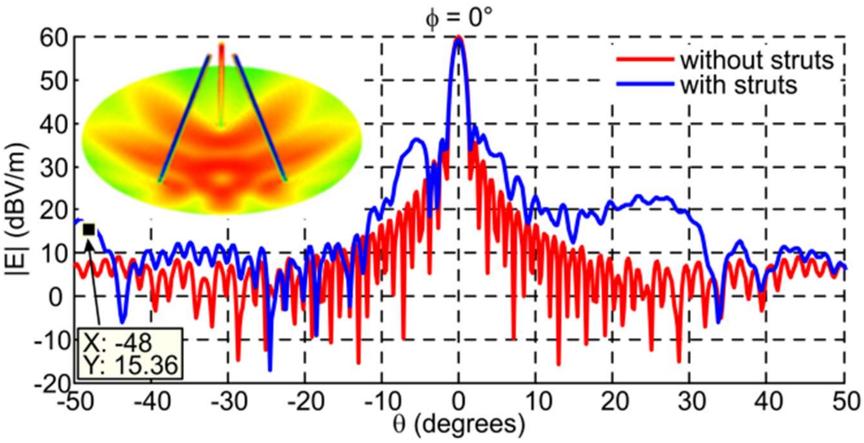


Fig. 4.27 – Nominal vs struts perturbed radiation pattern. The inset shows the IPO 4 iterations converged current distribution.

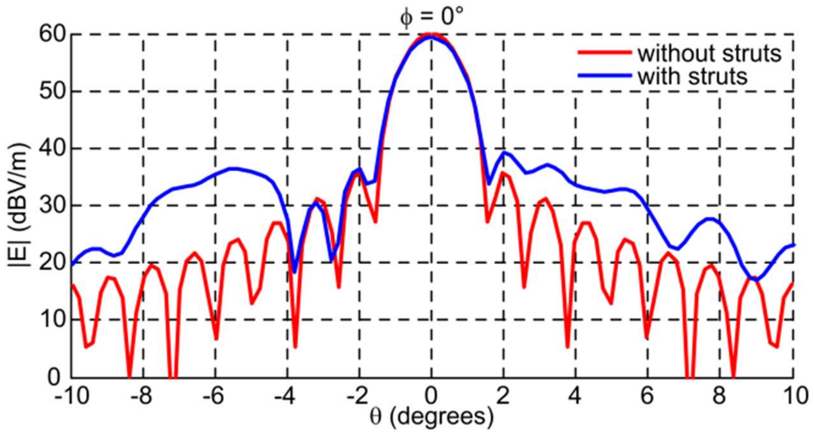


Fig. 4.28 – Nominal vs struts perturbed radiation pattern: zoom in the main beam and the near-in side-lobes.

4.7.5.1.2 Compact Antenna Test Range with Serrated Edges

This section provides the results for a dual parabolic cylindrical reflectors system, employed as a Compact Test Range (CTR), and these are compared against a full-wave analysis.

The system is specifically designed to operate at low frequencies down to 2 GHz and at very high frequencies up to 100 GHz. The compact range reflectors have an electrical size of about 1200λ by 967λ for a frequency of 100 GHz. These dimensions allow full-wave simulations only in the lower frequency region. Asymptotic methods like IPO need of significantly lower memory requirements and have the advantage of becoming more accurate the larger the structures are.

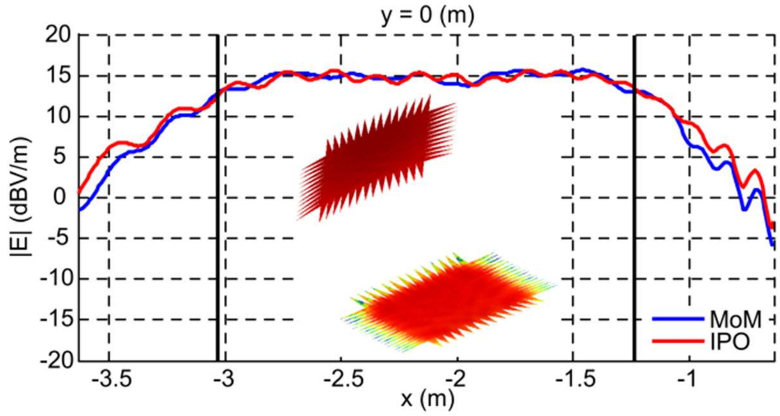


Fig. 4.29 – Amplitude of the electric field scattered by the CTR on the QZ as predicted by the IPO (red line) and the MoM (blue line) in the y = 0 m cut. The inset shows IPO induced currents at 4 GHz

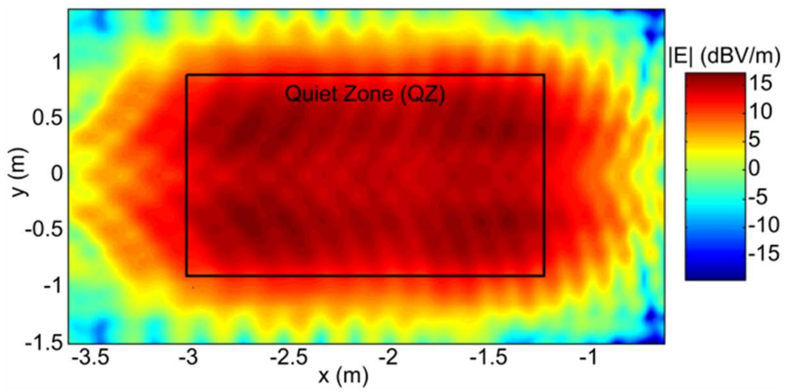


Fig. 4.30 – Amplitude of the electric field scattered by the CTR in the QZ predicted by the IPO

The simulation procedure is a simple "one-shot" procedure in which the complete antenna system, including reflectors, serrations, etc., is modeled by triangular/quadrangular facets. Feeders are typically represented by EM equivalent models as input (typically Spherical Wave Expansion, or electric and magnetic equivalent currents). Facilities for monitoring the convergence behaviour of IPO are available through different diagnostic parameters: local residual error on each facet of the structure; global residual error over the structure for each iterative step; structure currents induced on the model as the main output of the iterative procedure (inset in Fig. 4.29).

Some results evaluated at 4 GHz using a HP Z800 Workstation Intel® Xeon® CPU X5672 @ 3.20 GHz 8 cores with 96 GB RAM are reported for both IPO and MoM-MLFMA algorithms to evaluate IPO accuracy. The near field on the Quiet Zone (QZ) is evaluated at first (Fig. 4.29 and Fig. 4.30). The size of the QZ is 1.8 m x 1.8 m meters, which corresponds to the range bounded by square/lines in Fig. 4.30, where the amplitude of the electric field evaluated by IPO is shown. Fig. 4.29 shows the curves for the y = 0 m cut both for IPO and MoM-MLFMA.

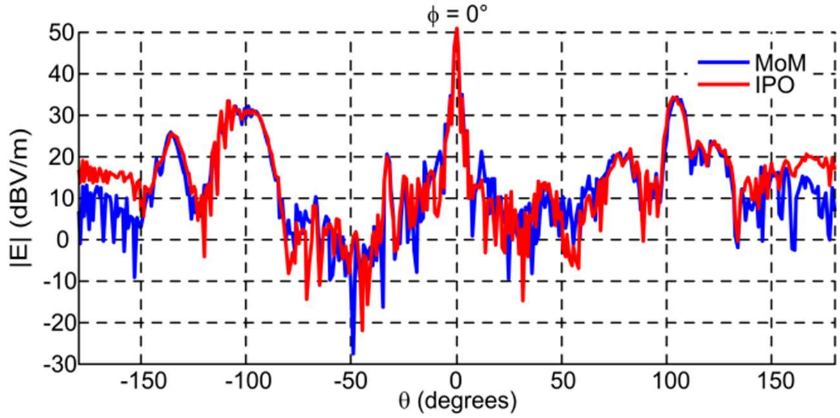


Fig. 4.31 – Amplitude of the electric far field scattered by the CTR predicted by IPO (red line) and MoM (blue line) along ϕ = 0° cut

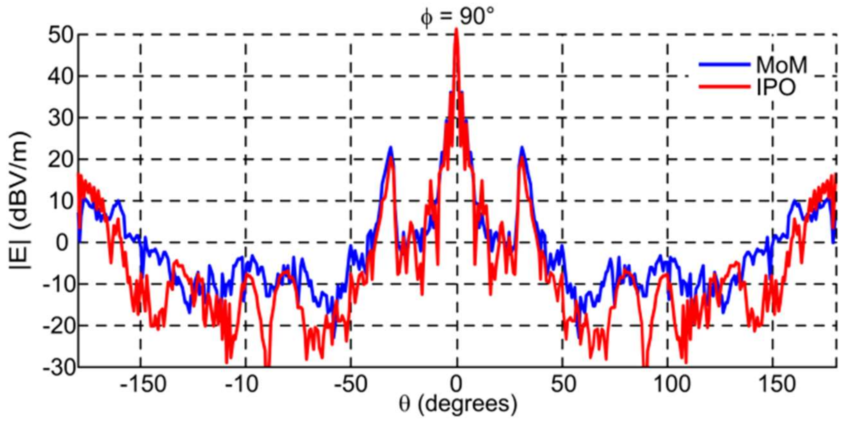


Fig. 4.32 – Amplitude of the electric far field scattered by the CTR predicted by IPO (red line) and MoM (blue line) along ϕ = 90° cut

MoM-MLFMAs				
N° facets	*N° threads*	*RAM*	*Elapsed Time*	*N° iterations*
493208	8	12 GB	15 m	30
IPO				
N° facets	*N° threads*	*RAM*	*Elapsed Time*	*N °iterations*
89381	8	0.2 GB	3 m	3

Tab. 4-6 – Computational Performance

Patterns along ϕ = 0° and ϕ = 90° cuts for both IPO and MoM solutions are compared in Fig. 4.31 and Fig. 4.32, respectively. The details of the computational performance are summarized in Tab. 4-6. In the proposed example, the IPO algorithm gives results that compare well with the full-wave solution. Moreover, it is easy to use; it significantly reduces the memory requirements and is very efficient in computational time, thus allowing CTR accurate analysis at frequencies where full-wave methods require unavailable computational resources.

4.7.5.2 Radomes

Assessment of radome quality requires the verification of performance in a significant number of antenna conditions, as stated in DO-213 [BD43]: transmission efficiency, side-lobe level, incident reflection, beam deflection, beam width. A significant degree of accuracy is required to be able to discriminate between the different classes of quality reported in [BD43]. However, the large electrical dimension of the radome sometimes limits the applicability of full-wave methods. The implemented IPO algorithm is particularly suitable to such kinds of applications, thanks to its capability to manage “partially transparent materials” and “multi-bounces”, while having a much lower computational cost than full-wave methods, thus removing analysis limitations due to the electrical dimensions. Note that by implementing an “integral current based approach”, IPO is also much better than ray-based techniques in terms of accuracy.

In particular, in this section, we investigate the impact of an airborne dielectric radome on the radiation pattern of a radar antenna operating in the X band (9.375 GHz). The geometrical model of the radome is obtained as a portion of the A-139 helicopter radome whose CAD model is shown in the inset in Fig. 4.33. The values of permittivity and thickness of the considered monolithic radome are $\varepsilon r = 3.85$ and $h = 1.58$ mm, respectively. The antenna and its support are depicted in yellow in the inset of Fig. 4.33. The antenna radiates a boresight beam in the helicopter front direction.

The EM interaction between the antenna and the radome are evaluated by the IPO solver taking into account multiple reflections and both parallel and perpendicular polarizations. The antenna is modeled as an equivalent current distribution that reproduces the measured radiation patterns, especially in the region corresponding to the main lobe and the first secondary lobes.

The free space radiation pattern of the synthesized antenna has the following features: maximum gain ≈ 28 dB; half-power beamwidth ≈ 8°; side lobe level ≈ 25 dB; circular opening (diameter 30 cm ≈ 10λ); and linear polarization.

Radomes can cause high side-lobes in radar-antenna patterns, which will increase clutter, false alarm rate, and susceptibility to jamming. The radomes can also cause deflection and attenuation of the main beam, filling of the difference-pattern null used for tracking, and interferometry errors. The IPO solver can help to predict these effects by providing the following performance parameters: incidence angles on the radome surface; power density on the radome surface; parallel and perpendicular components of the transmission coefficients on the radome; directivity of the antenna with and without radome (Fig. 4.33).

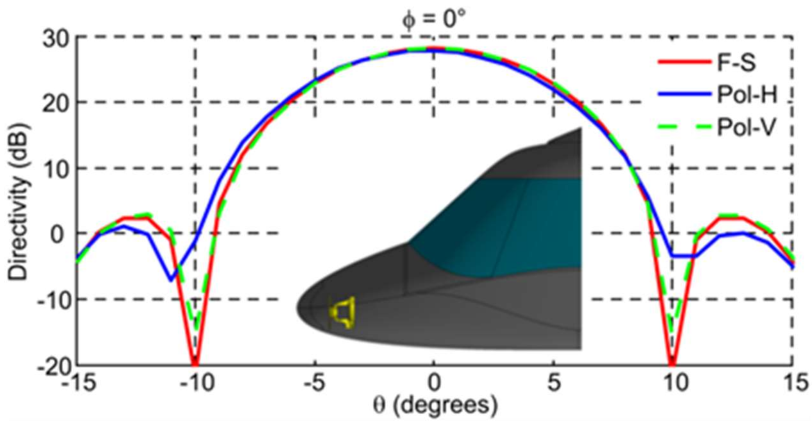


Fig. 4.33 – Directivity for different antenna polarizations on the principal cut in presence of the dielectric radome (blue and green lines). The Free-Space (F-S) pattern of the antenna in the absence of radome (red line) is also plotted as reference. The inset shows the radome shape and antenna position (in yellow).

Another application example can be the installation of a radar mounted on a metal bracket behind the bumper of a car. The bumper can causes ripples, deflection and attenuation of the main beam.

In the case studied, the pattern distortion of an anti-collision radar operating at 24 GHz was computed using IPO. The mesh model consists of 240025 facets corresponding to about 16 facets per square wavelength. The IPO converges in 4 iterations providing the current distribution is shown in the inset of Fig. 4.34. In addition, the radiation pattern of the radar in its operating condition is calculated (blue line) and directly compared against its accelerate versions (green and black lines) and the MoM result (Fig. 4.34).

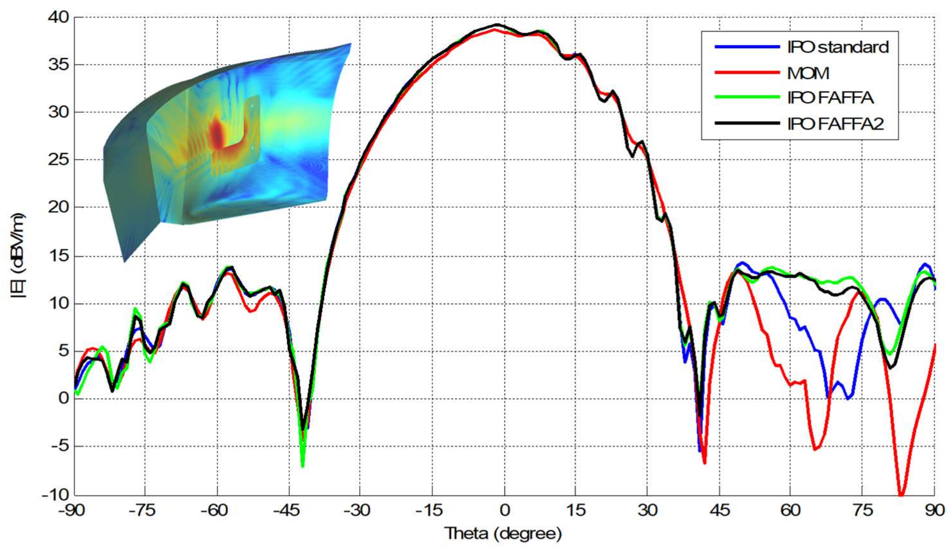


Fig. 4.34 – Anti-collision radar pattern when the antenna is installed behind the bumper of a car (blue). The IPO standard result (blue) is compared against its accelerated versions (green and black) and the MoM result (red). The inset shows the IPO current distribution on the satellite.

Method of Moments (MLFMA) - "one-shot" approach					
N° facets	***N° threads***	***RAM***	***N° iter***	***Elapsed Time***	
5512043	36	157GB	291 (res 10-4)	6h 22m	
Iterative Physical Optic @ 24GHz CPU					
N° facets	***N° threads***	***RAM***	***N° iter***	***Elapsed Time***	
240025	16	454 MB	4 (res.4E-02)	3h 10m	standard
240025	16	162 MB	4 (res.4E-02)	25m	FAFFA2

Tab. 4-7 – Computational Performance

Details of the computational performance are summarized in Tab. 4-7. In the proposed example, the IPO algorithm gives results that compare well with the full-wave solution (MoM). Moreover, it is easy to use; it significantly reduces the memory requirements and is very efficient in terms of computational time compared to MoM (~15x speedup).

4.7.5.3 Co-Site

Antenna placement aboard platforms (e.g., ships, aircrafts, satellites, cars, etc.) requires verification of a number of issues related both to performance (e.g., antenna coverage) and ElectroMagnetic Compatibility (EMC) (e.g., inter-antenna coupling, near-field hazard, radiated emission/susceptibility, etc.). Depending on the antenna working frequency and platform geometrical dimensions, several modeling techniques are usually applied: from full-wave methods in the low-frequency range (e.g., MoM, FDTD, etc.) up to ray-based techniques (e.g., UTD, SBR, etc.) in the upper frequency range. As happens in other application fields, an intermediate frequency range usually exists in which full-wave methods can no longer be applied due to the computational cost exceeding the available computational resources and where ray-based methods suffer from problems of applicability and accuracy. IPO method can again fill this gap, by preserving many of the desirable properties of the full-wave methods: e.g., detailed geometry representation, ease of use, multi-bouncing, integral current-based representation.

This section deals with the problem of the modification of the antenna pattern due to antenna interaction with a space-platform on which it has been installed; i.e., how the spacecraft body, the appendages, and the surrounding antenna systems (considered as passive structures) modify the far field antenna pattern and, consequently, its projection on Earth. In particular, the pattern distortion of a helix antenna operating at 2 GHz was computed using IPO. The mesh model consists of 234165 facets corresponding to about 16 facets per square wavelength. The IPO converges in 4 iterations providing the current distribution shown in the inset of Fig. 4.35. In addition the radiation pattern of the helix antenna in its operating condition is calculated and directly compared against its free-space pattern (Fig. 4.35).

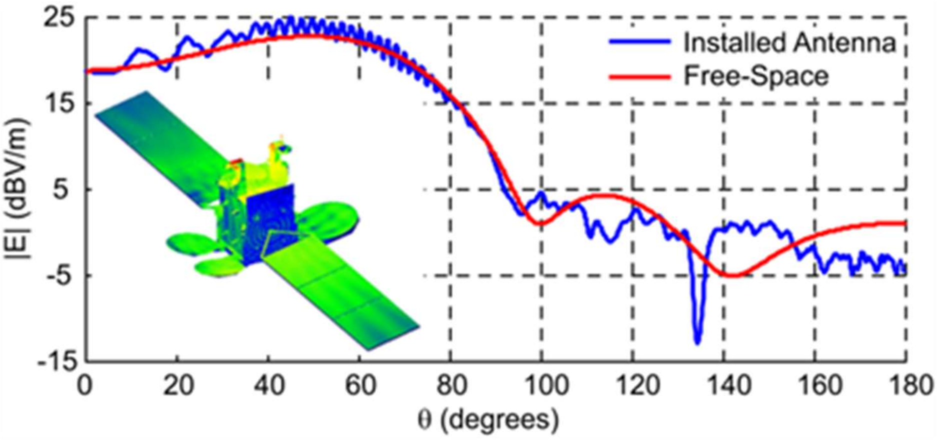
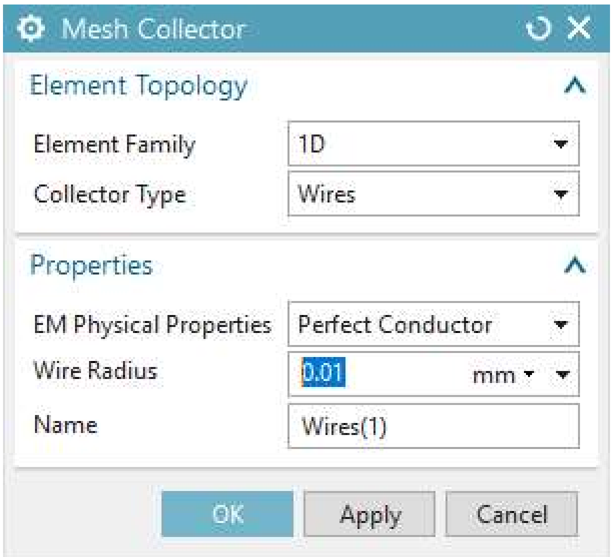


Fig. 4.35 – Helix antenna nominal free-space radiation pattern (red) and radiation pattern when the antenna is installed on board the satellite (blue). In the inset the IPO current distribution on the satellite is shown.

5. *REFERENCE GUIDE*

5.1 Simcenter 3D High Frequency EM Mesh Collector dialog box



<table>
<tr><th colspan="2">Element Topology</th></tr>
<tr><td>Element Family</td><td>Specifies the element family. Choose from:
1D
2D</td></tr>
<tr><td>Collector Type</td><td>Specifies the collector type for the selected family.

The element names depend on the solver language and on the analysis type chosen for the FE model.

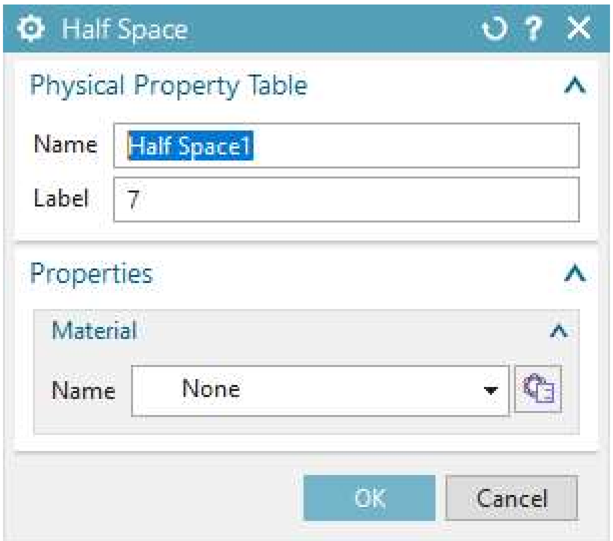
The following types are available for Simcenter 3D High Frequency EM FE Models:
3D MoM, 2.5D MoM and S-PEEC analysis types:
<table>
<tr><th>Collector Type</th><th>Element family</th><th>Element type</th></tr>
<tr><td rowspan="2">Electromagnetic shell</td><td>2D</td><td>Triangle</td></tr>
<tr><td>1D</td><td>Wire</td></tr>
<tr><td>EM Field probes</td><td>2D</td><td>EM Field probe</td></tr>
</table>
Iterative PO and UTD analysis types:
<table>
<tr><th>Collector Type</th><th>Element family</th><th>Element type</th></tr>
<tr><td rowspan="2">Electromagnetic shell</td><td rowspan="2">2D</td><td>Triangle</td></tr>
<tr><td>Quad</td></tr>
<tr><td>EM Field probes</td><td>2D</td><td>EM Field probe</td></tr>
</table>
</td></tr>
</table>

	The *EM Field Probe* type is used to define the surface where the output of near field on conformal scan analyses is computed. Other types are used for the mesh model to be analyzed.
Properties	
EM Physical Properties	Appears if **Collector Type** is **Electromagnetic shell**. Defines the material EM properties to be assigned to the elements in the mesh collector. Possible choices are: • **Perfect Conductor** • **Dielectric Boundary** (3D MoM Analysis Type only – 2D elements only) • **Aperture** (2.5D MoM – 2D elements only) • **Other Physical Material**
Type	Appears if **EM Physical Properties** is **Other Physical Material.** Specifies the type of physical property to be used to define material data for the collector mesh elements. Possible choices are: • **Half Space** (Iterative PO, UTD Analysis types only) • **Grid** • **Surface Impedance** • **Tabulated Data**
Physical Property	Appears if **EM Physical Properties** is **Other Physical Material.** This allows the selection or creation of a Physical Property of the type selected in **Type.**
Wire Radius	Appears if **Collector Type** is **Electromagnetic shell** and **Element Family** is **1D.** Specifies the radius to be used in the simulation for the collector 1D elements. Note: available in 3D MoM, 2.5D MoM and S-PEEC analyses only.
Name	Specifies the name of the mesh collector. Enter a meaningful name or accept the default.

Tab. 5-1 – "Simcenter 3D High Frequency EM Mesh Collector" GUI fields

5.2 Physical Property Tables

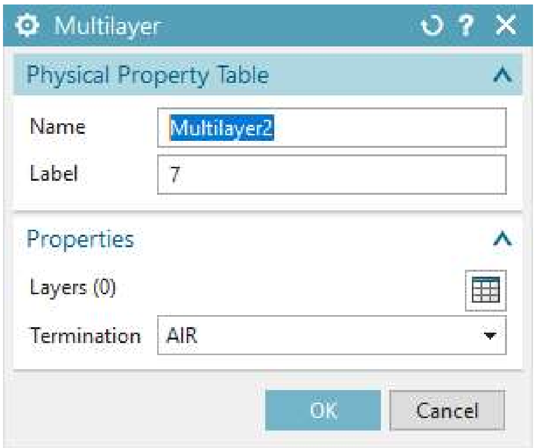
5.2.1 Half Space

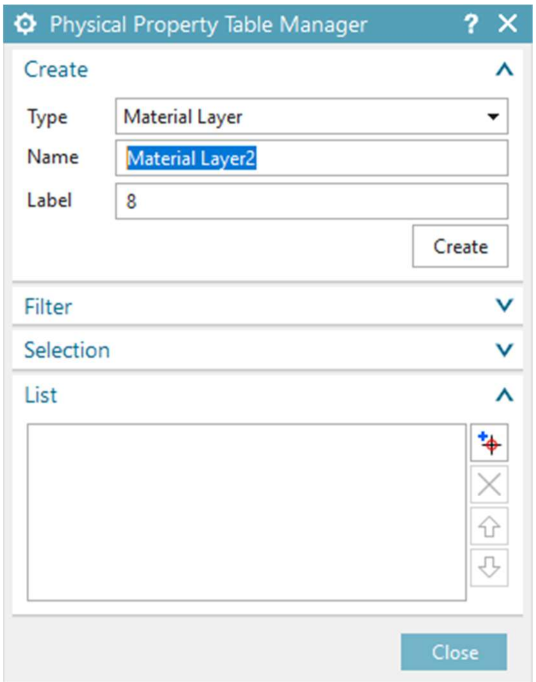


Physical Property Table	
Name	Defines the object name.
Label	Specifies a unique numerical identifier for the object.
Properties	
Material	Selects an isotropic material with high frequency electromagnetic properties.

Tab. 5-2 – "Half Space" GUI fields

5.2.2 Multilayer

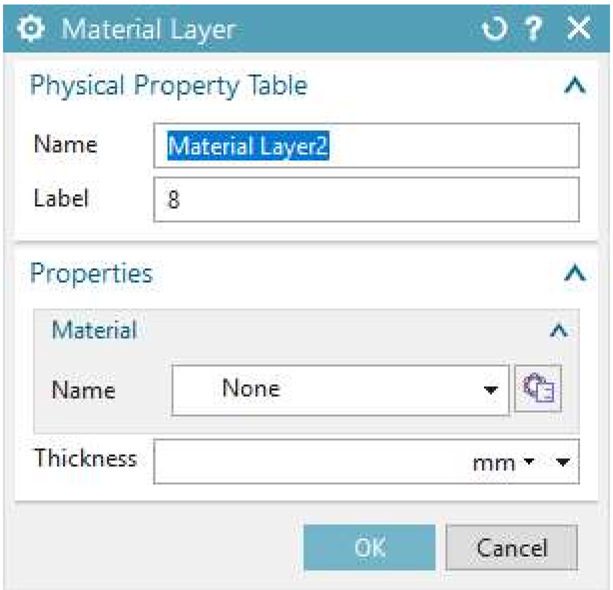




Physical Property Table	
Name	Defines the object name.
Label	Specifies a unique numerical identifier for the object.
Properties	
Termination	Specifies the multilayer termination material. Available options are: • Air • Perfect Electric Conductor (PEC)
Create	
Create	Defines the *Material Layer* physical property table by building the material.
Layers table	Shows the list of material layers, allowing you to edit, move, copy and remove them.

Tab. 5-3 – "Multilayer" GUI fields

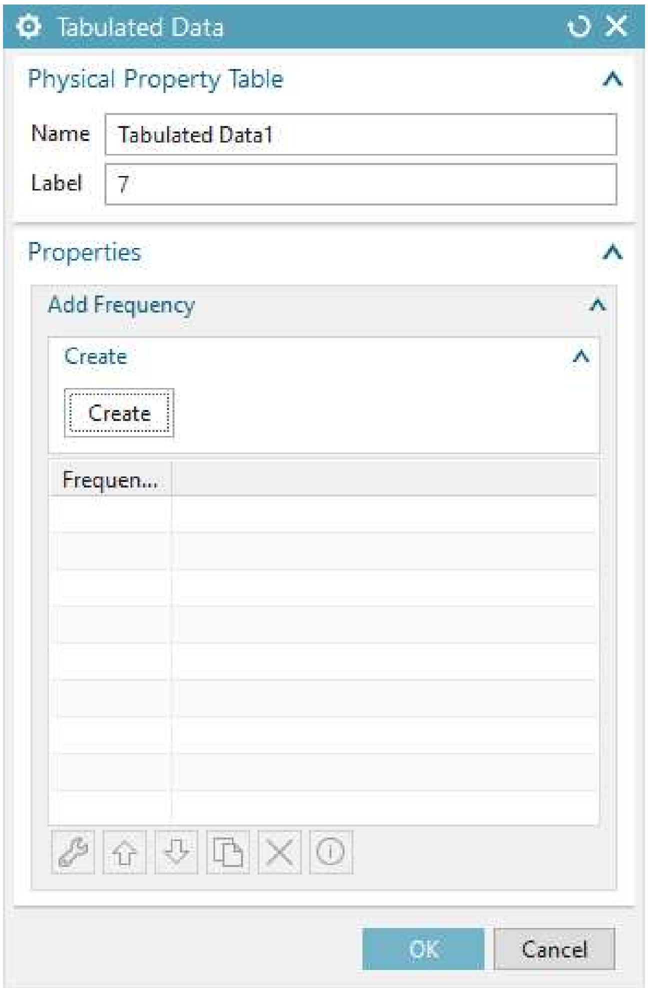
5.2.3 Material Layer



Physical Property Table	
Name	Defines the object name.
Label	Specifies a unique numerical identifier for the object.
Properties	
Material	Assigns an isotropic material with high frequency electromagnetic properties to the layer.
Thickness	Sets the layer thickness.

Tab. 5-4 – "Material Layer" GUI fields

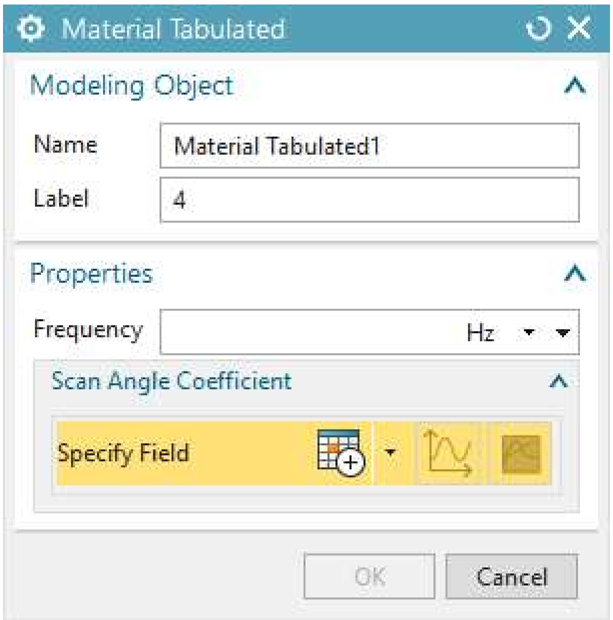
5.2.4 Tabulated Data



Physical Property Table	
Name	Defines the object name.
Label	Specifies a unique numerical identifier for the object.
Add Frequency	
Create	Creates a *Material Tabulated* physical property table which defines material parameters for a given frequency.
Frequencies table	Specifies the frequency dependent material parameters as the list of *Material Tabulated* physical property table.

Tab. 5-5 – "Tabulated Data" GUI fields

5.2.5 Material Tabulated

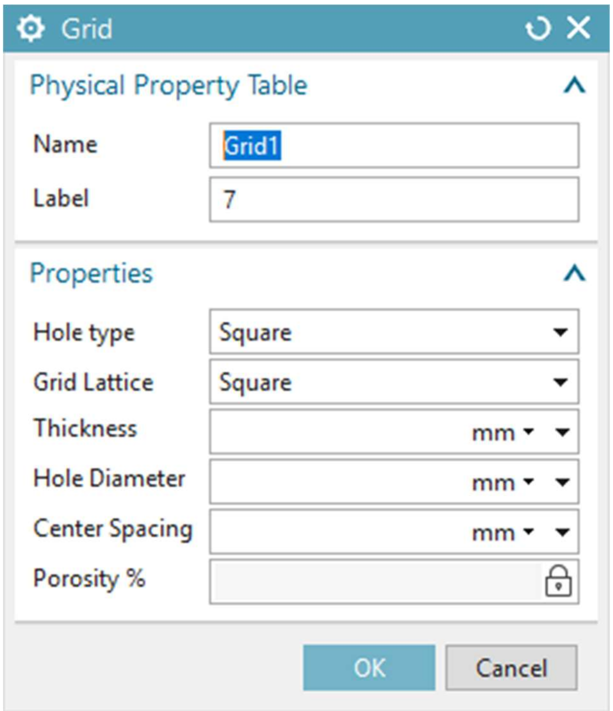


Physical Property Table	
Name	Defines the object name.
Label	Specifies a unique numerical identifier for the object.
Properties	
Frequency	Specifies the frequency related to the material parameters defined in this object.
Scan Angle Coefficient	Defines a field where material parameters are specified as a function of the incidence angle. Field parameters are: ➢ **Angle** is the incidence angle (independent variable); ➢ **R // (TM) Real** is the real part of the reflection coefficient for the parallel polarisation; ➢ **R // (TM) Imaginary** is the imaginary part of the reflection coefficient for the parallel polarisation; ➢ **R _|_ (TE) Real** is the real part of the reflection coefficient for the perpendicular polarisation; ➢ **R _|_ (TE) Imaginary** is the imaginary part of the reflection coefficient for the perpendicular polarisation; ➢ **T // (TM) Real** is the real part of the transmission coefficient for the parallel polarisation;

	➢ **T // (TM) Imaginary** is the imaginary part of the transmission coefficient for the parallel polarisation; ➢ **T _|_ (TE) Real** is the real part of the transmission coefficient for the perpendicular polarisation; ➢ **T _|_ (TE) Imaginary** is the imaginary part of the transmission coefficient for the perpendicular polarisation.

Tab. 5-6 – "Material Layer" GUI fields

5.2.6 Grid

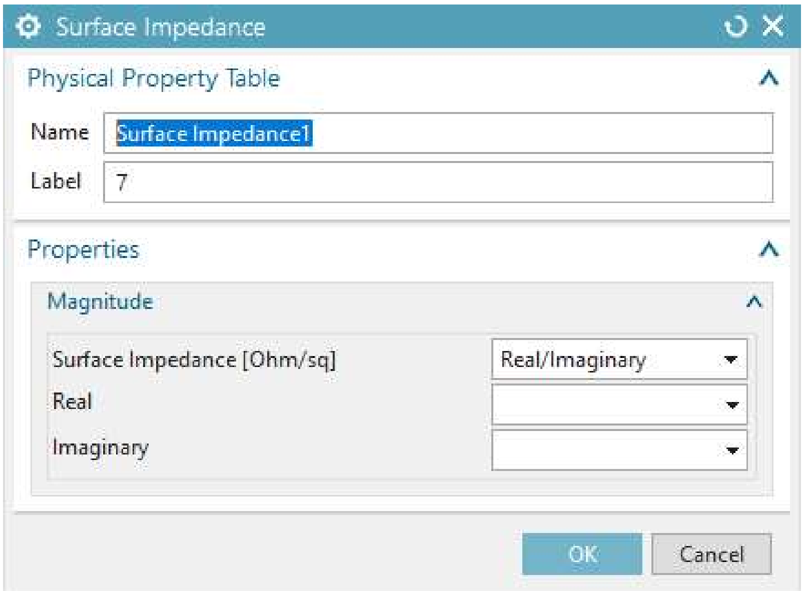


Physical Property Table	
Name	Defines the object name.
Label	Specifies a unique numerical identifier for the object.
Properties	
Hole type	Specifies the grid hole shape. This can be: • **Square** • **Circular**
Grid Lattice	Specifies the grid lattice shape. This can be: • **Square** • **Triangular**

Thickness	Specifies the grid thickness.
Hole Diameter	For circular grid holes, specifies the hole diameter. For square grid holes, specifies the length of the side of the square.
Center Spacing	Specifies the minimum distance between two hole centers.
Porosity %	This is a read-only field and it is automatically calculated as the ratio of the void to the total surface of the grid.

Tab. 5-7 – "Grid" GUI fields

5.2.7 Surface Impedance

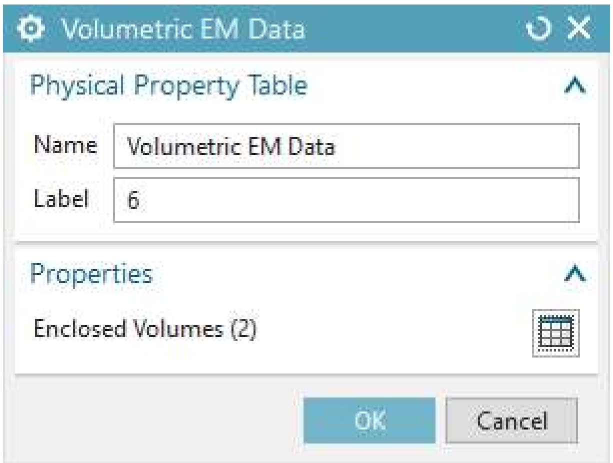


Physical Property Table	
Name	Defines the object name.
Label	Specifies a unique numerical identifier for the object.
Properties	
Surface Impedance	Specifies how the impedance complex value is defined. Available options are: • Real/Imaginary • Magnitude/Phase
Real	Appears when **Surface Impedance** is Real/Imaginary. Specifies the real part of the impedance.
Imaginary	Appears when **Surface Impedance** is Real/Imaginary.

	Specifies the imaginary part of the impedance.
Magnitude	Appears when **Surface Impedance** is Magnitude/Phase. Specifies the magnitude of the impedance.
Phase	Appears when **Surface Impedance** is Magnitude/Phase. Specifies the phase of the impedance.

Tab. 5-8 – "Surface Impedance" GUI fields

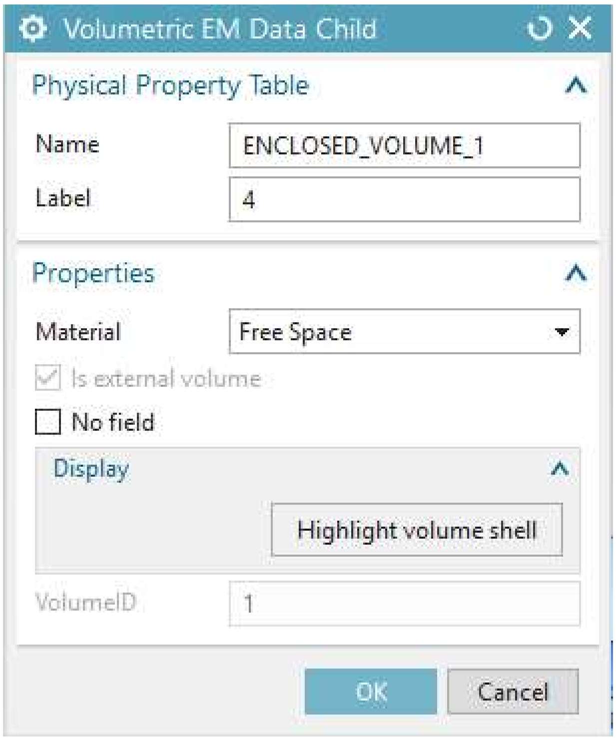
5.2.8 Volumetric EM Data



Physical Property Table	
Name	Defines the object name. Note: this type of physical property table is only used in *3D MoM* analyses.
Label	Specifies a unique numerical identifier for the object.
Properties	
Enclosed Volumes	Opens the list of volumes, where you can edit them. Items can also be moved up and down in the list but their position doesn't affect any function. This list must remain consistent with the mesh so copy and delete functions are available but these types of changes can be applied only if the final list of volumes contains the same number of elements with the same volume IDs as the original list. Note: Full-wave Ports and Lumped Impedance (Regions) cannot be placed on the geometry/mesh dividing two different volumes.

Tab. 5-9 – "Volumetric EM Data" GUI fields

5.2.9 Volumetric EM Data Child

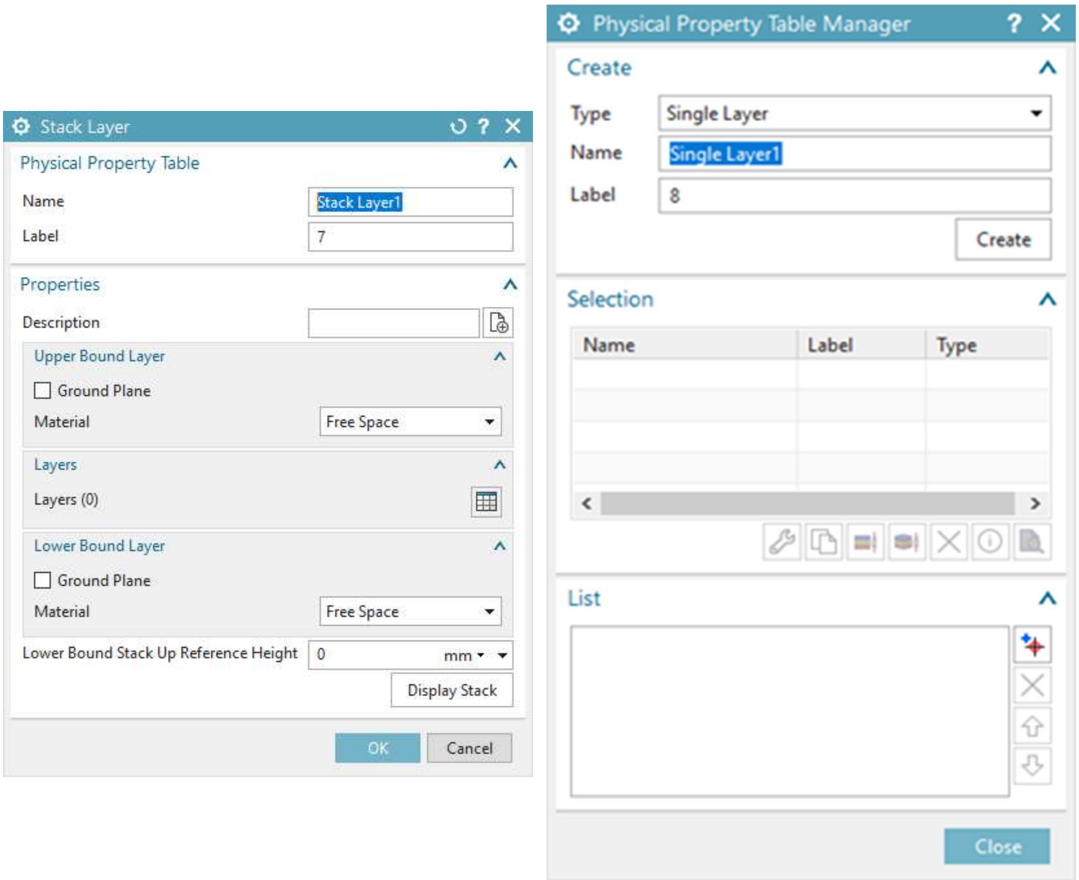


Physical Property Table	
Name	Defines the object name. Note: This type of physical property table is only used in *3D MoM* analyses.
Label	Specifies a unique numerical identifier for the physical property table.
Properties	
Material	Defines the material filling the enclosed volume. This can be a physical material or free space (default value).
Physical Material	Appears when **Material** is set to **Physical material**. Assigns an isotropic material with high frequency electromagnetic properties to the enclosed volume.
Is external volume	This is an automatically created read-only informative field . It is false for all the closed domains and true for the external volume only.
No field	This is set if the electromagnetic field inside the volume is forced to zero by the solver. Default value is True if the volume shell is a Conductor (Perfect Electric Conductor).

Volume ID	This is an automatically created read-only informative field This value is used internally by the solver.
Display	
Highlight volume shell	Highlights the mesh elements forming the volume boundary.

Tab. 5-10 – “Volumetric EM Data Child” GUI fields

5.2.10 Stack Layer

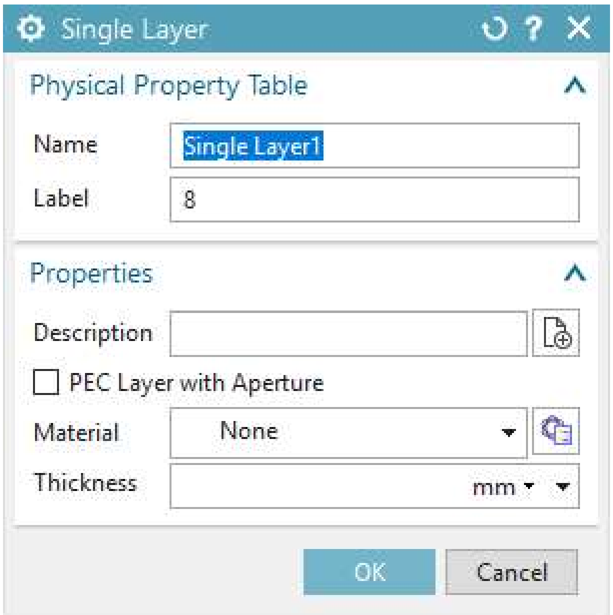


Physical Property Table	
Name	Defines the object name. <u>Note</u>: this type of physical property table is only used in *2.5D MoM* analyses.
Label	Specifies a unique numerical identifier for the physical property tables.
Properties	

Description	You can enter a brief description here. Click to open a text editor where you can enter a longer description.
Lower Bound Stack Up Reference Height	This field can be used to assign the ending Z coordinate of the **Lower Bound Layer**. Note: by default, the layer stack-up contains two layers: • ***"Lower Bound Layer"***, that describes the characteristic of the lower half space; its material is free space by default. • ***"Upper Bound Layer"***, that describes the characteristic of the upper half space; its material is free space by default.
Upper Bound Layer	
Ground Plane	Specifies if the upper bound layer is the ground plane. In this case, no material definition is required.
Material	Appears if **Ground Plane** option is not checked. Allows you to select the *Free Space* material or define/select a *Physical Material*, using the **Material List dialog box**.
Physical Material	Appears when **Material** is set to **Physical material**. Assigns an isotropic material with high frequency electromagnetic properties to the Upper Bound Layer.
Layers	
Create	Defines the *Single Layer* physical property table building the layer stack-up, between the lower and the upper bound layers.
Layers table	Shows the list of layers of the layer stack-up, allowing to edit, move, copy and remove them.
Lower Bound Layer	
Ground Plane	Specifies if the lower bound layer is the ground plane. In this case, no material definition is required.
Material	Appears if **Ground Plane** option is not checked. You can select the *Free Space* material or define/select a *Physical Material*, using the **Material List dialog box**.
Physical Material	Appears when **Material** is set to **Physical material**. Assigns an isotropic material with high frequency electromagnetic properties to the Upper Bound Layer.
Display	
Display Stack	This button lets you view a representation of the defined Stack Layer on the model,.

Tab. 5-11 – “Stack Layer” GUI fields

5.2.11 Single Layer

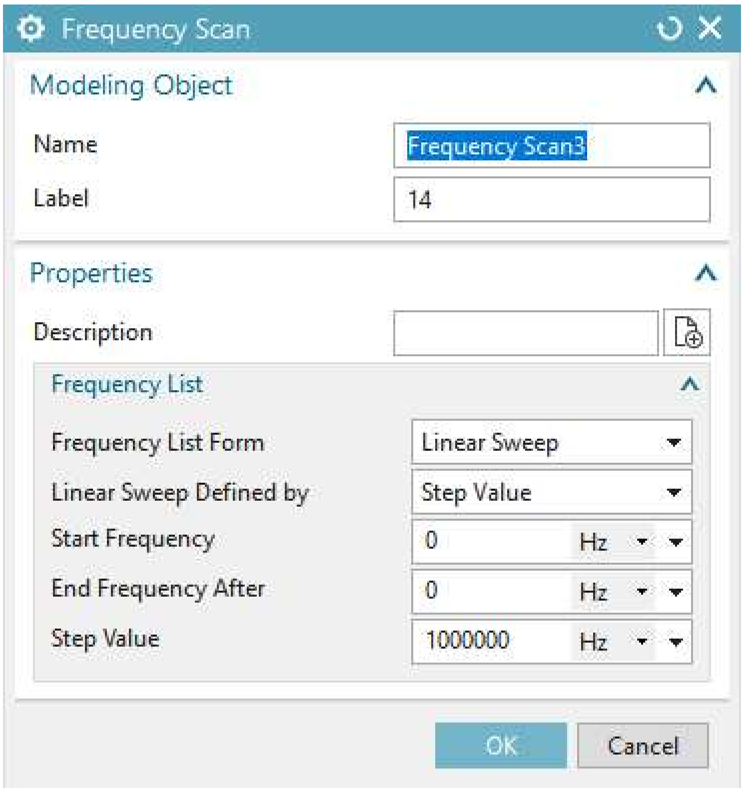


Physical Property Table	
Name	Defines the object name. Note: These types of modelling objects are used in *2.5D MoM* analyses only.
Label	Specifies a unique numerical identifier for physical property table.
Properties	
Description	You can enter a brief description here. Click to open a text editor where you can enter a longer description.
PEC Layer with Aperture	Specifies if the layer is an infinite ground plane on which apertures can be placed.
Material	Appears if **PEC Layer with Aperture** option is not checked. Assigns an isotropic material with high frequency electromagnetic properties to the layer.
Thickness	Appears if **PEC Layer with Aperture** option is not checked. Sets the layer thickness.

Tab. 5-12 – “Single Layer” GUI fields

5.3 Modeling Objects

5.3.1 Frequency Scan



Modeling Object	
Name	Defines the modeling object name.
Label	Specifies a unique numerical identifier for the modeling object.
Properties	
Description	You can enter a brief description here (optional). Click  to open a text editor where you can enter a longer description.
Frequency List	
Frequency List Form	Specifies the form of the frequency list. You can select the frequency list form from a list of individual frequencies, linear sweep or logarithmic sweep. • **Individual frequencies** definition: select the measurement unit and insert the list values; • **Linear sweep** definition: you can define the linear sweep in two ways:

<table>
<tr><td></td><td>
<li style="list-style:none">
by step value, inserting the start frequency, the end frequency after and the step value;
by number of steps, inserting the first frequency, the step value and the number of steps.

Logarithmic sweep definition: you can define the logarithmic sweep in two ways:

by interval, inserting the first frequency, the last frequency after and the number of logarithmic intervals;
by base, inserting the start frequency, the end frequency after and the base of logarithm.

</td></tr>
<tr><td>Frequency List</td><td>Appears when Frequency List Form is set to Individual Frequencies.

Specifies the frequency unit.</td></tr>
<tr><td>Frequency List box</td><td>Appears when Frequency List Form is set to Individual Frequencies.

You can specify as many frequencies as you need.

Enter the frequencies to include. Separate them by commas or spaces.

You can also Cut, Copy, and Paste the frequencies.</td></tr>
<tr><td>Linear Sweep Defined by</td><td>Appears when Frequency List Form is set to Linear Sweep.

Specifies the frequency range.

Number of Steps: you can set values for First Frequency, Step Value and Number of Steps.
Step Value: you can set values for Start Frequency, End Frequency After, and Step Value.

<u>Note</u>: when the End Frequency After value does not fit into the sweep definition, the software includes the next valid frequency in the frequency list.

For example, if Start Frequency is 100, End Frequency After is 200, and Step Value is 75, the software exports 100, 175, and the next valid frequency, which is 250.

When the End Frequency After value is greater than Start Frequency, the software generates one interval.

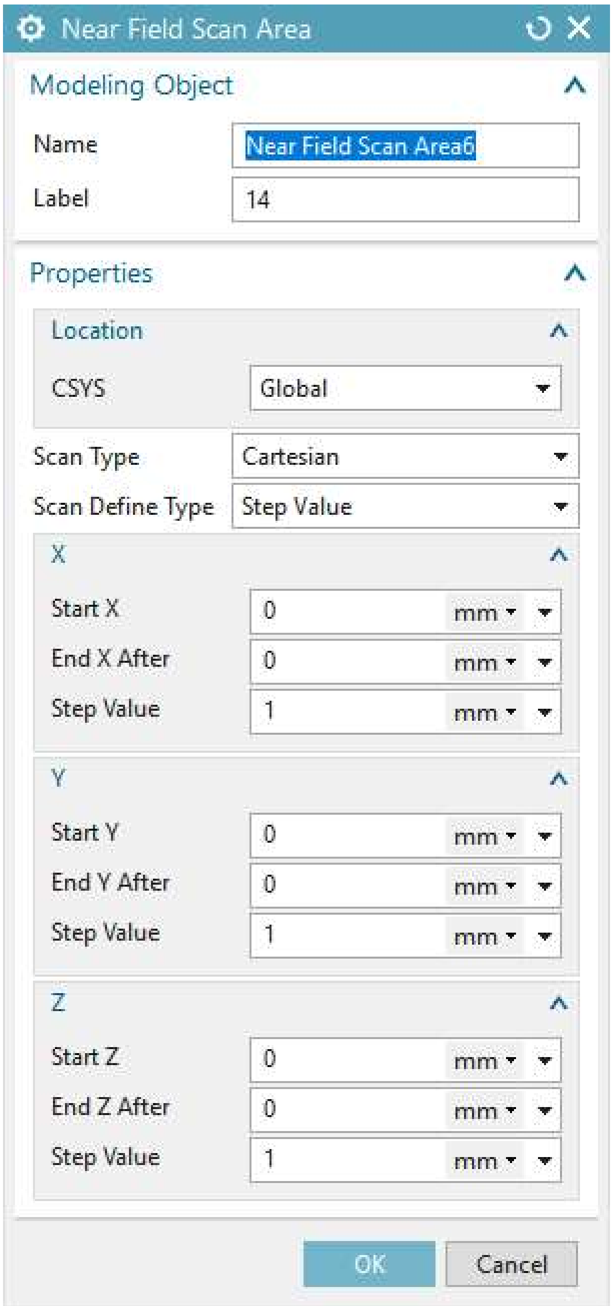
</td></tr>
<tr><td>Logarithmic Sweep Defined by</td><td>Appears when Frequency List Form is set to Logarithmic Sweep.

Specifies the frequency range.</td></tr>
</table>

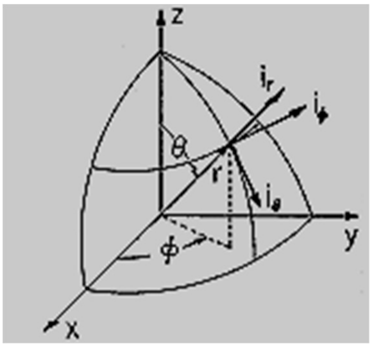
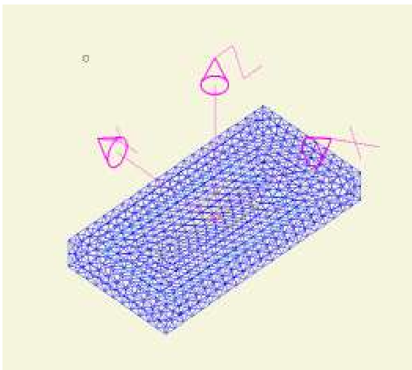
	• **Interval**: you can set values for First Frequency, Last Frequency and Number of Logarithmic Intervals. • **Base**: you can set values for Start Frequency, End Frequency After and Base of Logarithm. Note: when the End Frequency After value does not fit into the sweep definition, the software includes the next valid frequency in the frequency list. For example, if Start Frequency is 100, End Frequency After is 200, and Base of Logarithm is 75, the software exports 100, 175, and the next valid frequency, which is 250. When the End Frequency After value is greater than Start Frequency, the software generates one interval.

Tab. 5-13 – "Frequency Scan" GUI fields

5.3.2 Near Field Scan Area



Modeling Object	
Name	Defines the modeling object name.
Label	Specifies a unique numerical identifier for the modeling object.
Properties	
CSYS	The **CSYS** field lets you specify a coordinate system, in two different way:

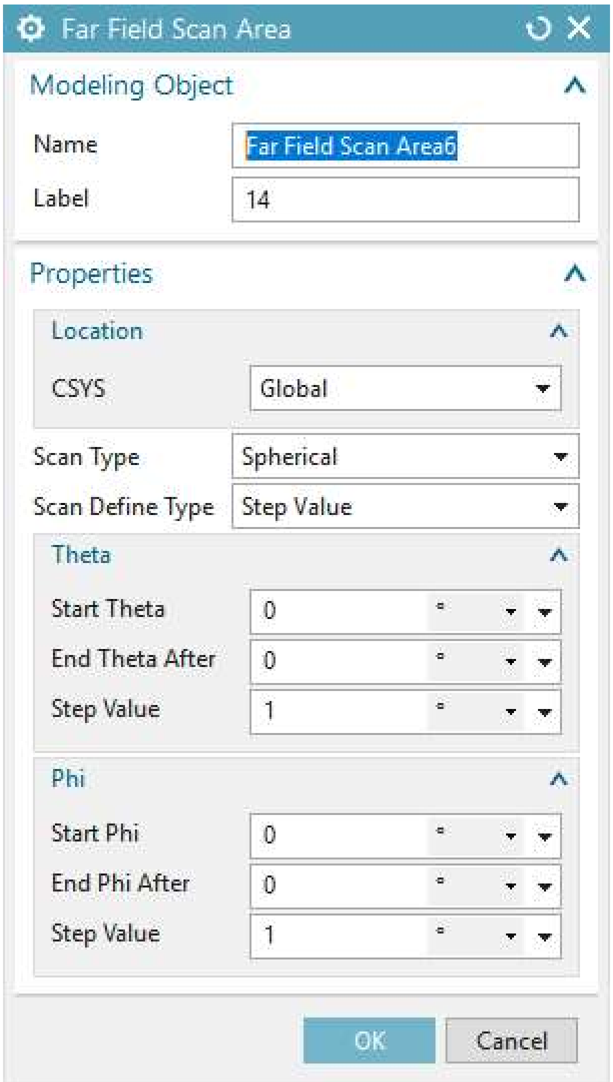
	• **Global**: scan area is defined with respect to the global coordinate system; • **User define**: specifies the scan area coordinate system using the **CSYS dialog box**.
Scan Type	Near field Scan Areas can be specified in three different coordinate systems (1.5.2): • **Cartesian**: the canonical x,y,z system reference; • **Spherical**: the coordinate system used is the following:  where ϕ is the angle with the X axis and θ is the angle with the Z axis. ϕ valid values are in the range (-360, 360) deg., while θ valid values are in the range (-180, 180) deg. *Rho* is the sphere radius. Note: a correct definition of the geometrical scan range should avoid redundancy in the angular ranges. • **Conformal**: is an arbitrarily shaped and ***meshed*** surface, usually but not necessarily, wrapped around a radiating device. Mesh elements to be used for meshing conformal scans shall be of *EM Field Probe* type. The following figure shows an example of a conformal scan defined for a patch antenna: 

Scan Define Type	Appears when **Scan Type** is set to **Cartesian** or **Spherical**. Specifies the scan area definition type: • **Number of Steps**: you can set, for each coordinate of the system, values for First point, Step Value and Number of Steps. • **Step Value**: you can set, for each coordinate of the system, values for Start point, End point After, and Step Value. Note: when the End point After value does not fit into the sweep definition, the software includes the next valid value in the list. For example, if Start point is 100, End point After is 200, and Step Value is 75, the software exports 100, 175, and the next valid value, which is 250. When the End point After value is greater than Start point, the software generates one interval.
Group reference	Appears when **Scan Type** is set to **Conformal**. Let's you use a group made of *EM Field Probe* elements to define a conformal scan. Select the group from the list, or click **New Group** to create a new group. Click **Edit Group** to open the **Edit Group** dialog box and access all commands related to groups.

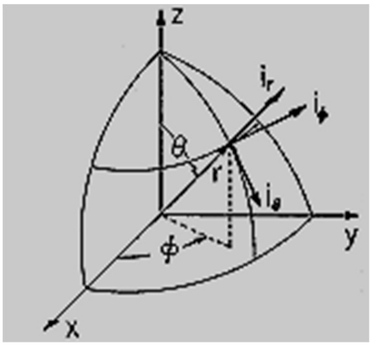
Scan definition	There are two ways to define the scan, based on the **Scan Type**:
	• If **Scan Type** is set to **Cartesian** or **Spherical**, you have to define the scan area limits:
	  ○ if Scan Type is **Cartesian**, define X, Y and Z sweep values;
	  ○ if Scan Type is **Spherical**, define Rho, Theta and Phi sweep values;
	• If **Scan Type** is set to **Conformal**, you have to select the surface that define the scan area, using the **QuickPick** or a group (see **Group reference**).

Tab. 5-14 – "Near Field Scan Area" GUI fields

5.3.3 Far Field Scan Area



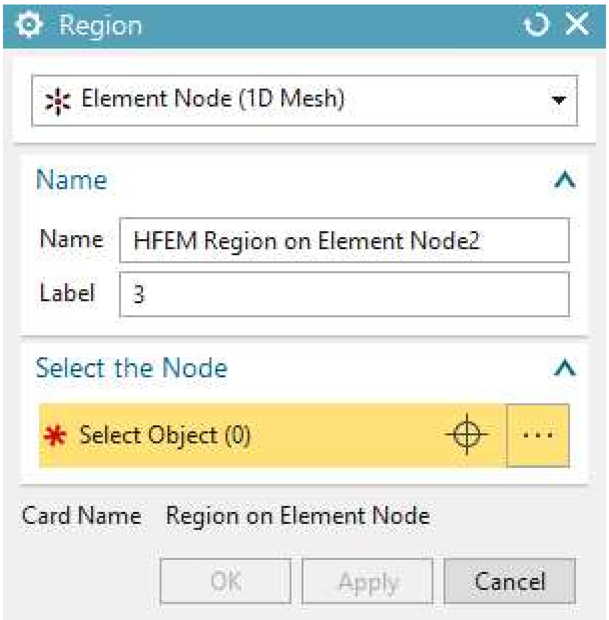
Modeling Object	
Name	Defines the modeling object name.
Label	Specifies a unique numerical identifier for the modeling object.
Properties	
CSYS	The **CSYS** field lets you specify a coordinate system, in two different way: • **Global**: scan area is defined with respect to the global coordinate system; • **User define**: specifies the scan area coordinate system using the [CSYS dialog box]().
Scan Type	Near field Scan Areas can be specified in two different coordinate systems (1.5.2):

	• **Spherical**: the coordinate system used is the following:  where ϕ is the angle with the X axis and θ is the angle with the Z axis. ϕ valid values are in the range (-360, 360) deg., while θ valid values are in the range (-180, 180) deg. <u>Note:</u> a correct definition of the geometrical scan range should avoid redundancy in the angular ranges. • **UV**: the U, V coordinate system is related to the spherical one through the following relations $u = \sin\vartheta\cos\varphi$ $v = \sin\vartheta\sin\varphi$
Scan Define Type	Specifies the scan area definition type: • **Number of Steps**: you can set, for each coordinate of the system, values for First point, Step Value and Number of Steps. • **Step Value**: you can set, for each coordinate of the system, values for Start point, End point After, and Step Value. <u>Note</u>: when the End point After value does not fit into the sweep definition, the software includes the next valid value in the list. For example, if Start point is 100, End point After is 200, and Step Value is 75, the software exports 100, 175, and the next valid value, which is 250. When the End point After value is greater than Start point, the software generates one interval.
Scan definition	
Coordinate definition	Appears when **Scan Type** is set to **UV** or **Spherical**. You can define the scan area limits, based on the Scan Type: • if Scan Type is **Spherical**, define Theta and Phi sweep values;

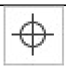
	• if Scan Type is **UV**, define U and V sweep values;

Tab. 5-15 – "Far Field Scan Area" GUI fields

5.4 Region dialog box (Simcenter 3D HF EM)



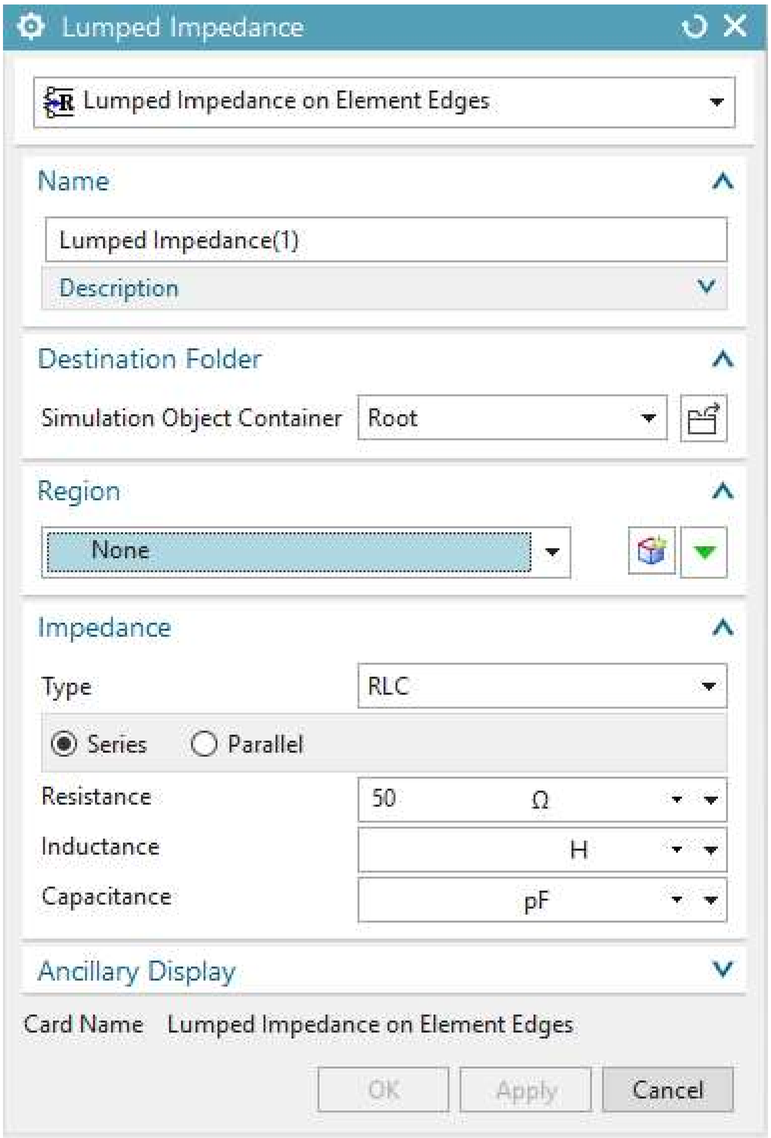
Region	
Region type list	Specifies the type of region to create: • **Element Edges (2D Mesh):** lets you define a collection of 2D element (triangle) edges. Note: the edges must be consecutive, i.e. they must have a common node. • **Element Node**: lets you select a node. • **Polygon Edge:** lets you select a single edge.
Name	
Name	Defines the region name.
Label	Specifies a unique numerical identifier for the region.
Select the Edge / Select the Node	
Group Reference	Let's you use a group to define a region. Select the group from the list, or click **New Group**  to create a new group. Click **Edit Group**  to open the **Edit Group** dialog box and access all commands related to groups.

Filter Type	Let’s you select the type of object in the group to use to define the region.
Select Object 	Let’s you select the edges, element edges or nodes to define a new region.
Polarity	
Automatic definition of Positive Side Elements	When this option is selected (default) you can select just the edges relevant to the region, the relative mesh elements (triangles) of the positive and the negative side will be automatically collected. If this option is not active, you have to select the edges, the elements of the positive group and the negative one. Note: this option is available only for “Element Edges (2D Mesh)” region type.

Tab. 5-16 – “Region” GUI fields

5.5 Simulation Objects

5.5.1 Lumped Impedance



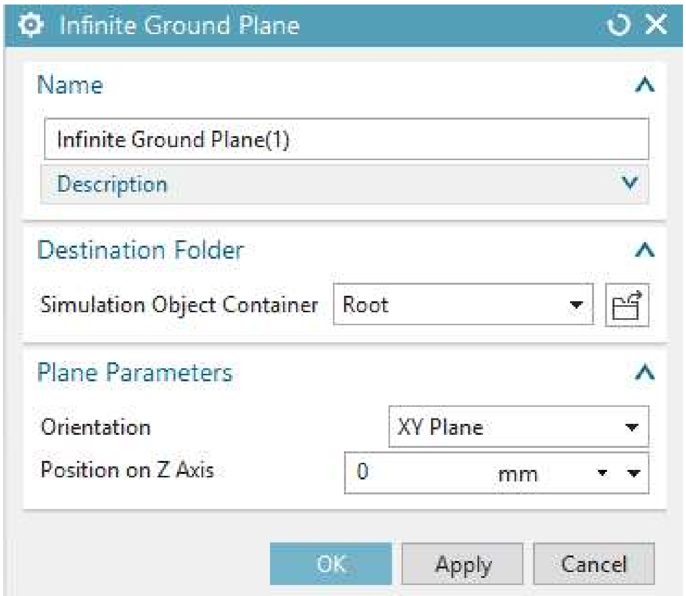
Simulation Object	
Region type list	You can choose the type of Region where the Lumped Impedance is defined; available choices are: • **Lumped Impedance on Element Edges** • **Lumped Impedance on Element Node** • **Lumped Impedance on Polygon Edge**
Name	Defines the object name. Note: These types of objects are used in *3D MoM, 2.5D MoM* and *S-PEEC* analyses only.

Description	You can enter a brief description here (optional).
Destination Folder	
Simulation Object Container	Specifies the container of the simulation object. Default is the root of Simulation Object Container.
Region	
Region selector	You can select or define a region to be assigned to the impedance, using the Create region tool. The available type of region depends on the selection in the **Region type list** field.
Impedance	
Type	Available types: • **RLC**: you can model the electrical impedance by a series or a parallel of a resistance, a capacitance and an inductance; • **Z**: you can model the electrical impedance by means of an impedance (specifying real and imaginary part); • **Y**: you can model the electrical impedance by means of an admittance (specifying real and imaginary part); • **External**: the electrical impedance is loaded from an external Touchstone file (.s1p or .s2p; if .s2p is used, the second port will be considered in short-circuit).
Series/Parallel	Appears when **Type** is set to **RLC**. Specifies if the impedance is a series or a parallel of a resistance, a capacitance and an inductance.
Resistance	Appears when **Type** is set to **RLC**. Defines (if needed) the resistance value of the impedance.
Inductance	Appears when **Type** is set to **RLC**. Defines (if needed) the inductance value of the impedance.
Capacitance	Appears when **Type** is set to **RLC**. Defines (if needed) the capacitance value of the impedance.
Impedance	Appears when **Type** is set to **Z** or **Y**. Specifies how the Z or Y complex value is defined. Available options are: • Real/Imaginary

	• Magnitude/Phase
Real	Appears when **Type** is set to **Z** or **Y** and **Impedance** is Real/Imaginary. Specifies the real part of the impedance.
Imaginary	Appears when **Type** is set to **Z** or **Y** and **Impedance** is Real/Imaginary. Specifies the imaginary part of the impedance.
Magnitude	Appears when **Type** is set to **Z** or **Y** and **Impedance** is Magnitude/Phase. Specifies the magnitude of the impedance.
Phase	Appears when **Type** is set to **Z** or **Y** and **Impedance** is Magnitude/Phase. Specifies the phase of the impedance.
Touchstone file	Appears when **Type** is set to **External**. You can select the external impedance Touchstone file (.s1p or .s2p; if .s2p is used, the second port will be considered in short-circuit).

Tab. 5-17 – "Lumped Impedance" GUI fields

5.5.2 Infinite Ground Plane



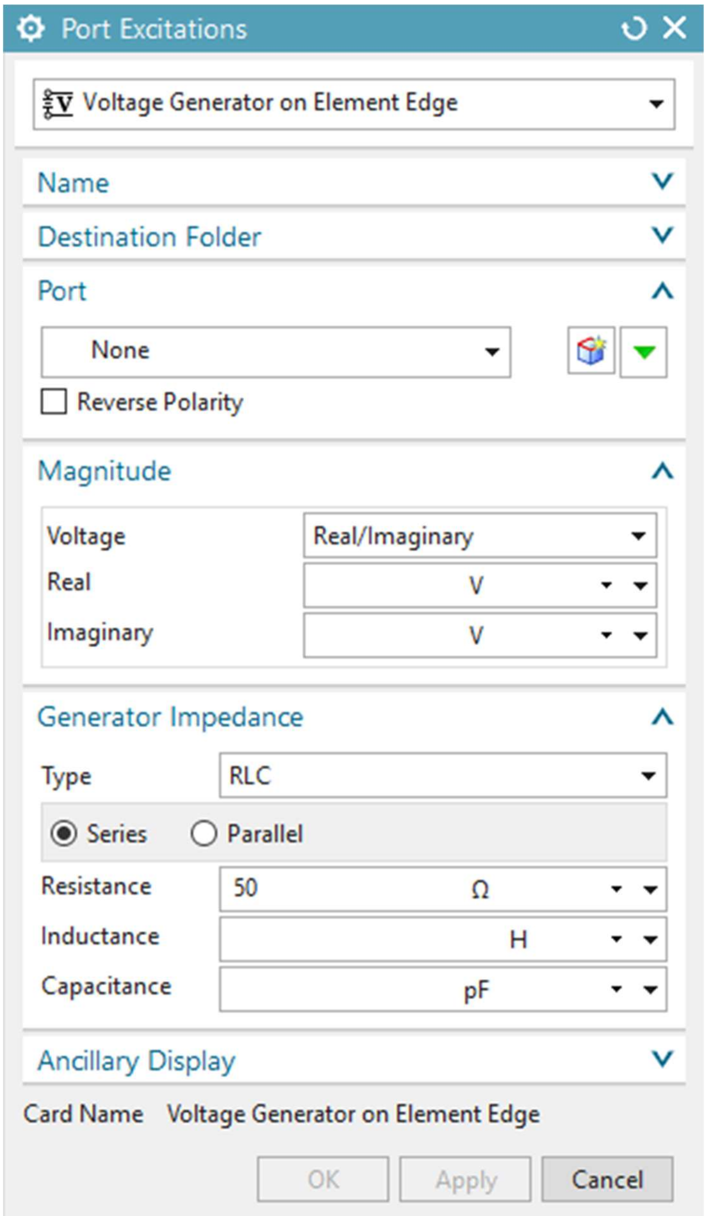
Simulation Object	
Name	Defines the object name.
Description	You can enter a brief description here (optional).
Destination Folder	

Simulation Object Container	Specifies the container of the simulation object. Default is the root of Simulation Object Container.
Plane Parameters	
Orientation	Only the **XY Plane** can be selected.
Position on Z Axis	Specifies the Z quota of the ground plane position. Note: zero is the only valid value for this field.

Tab. 5-18 – “Infinite Ground Plane” GUI fields

5.6 Loads

5.6.1 Port Excitations



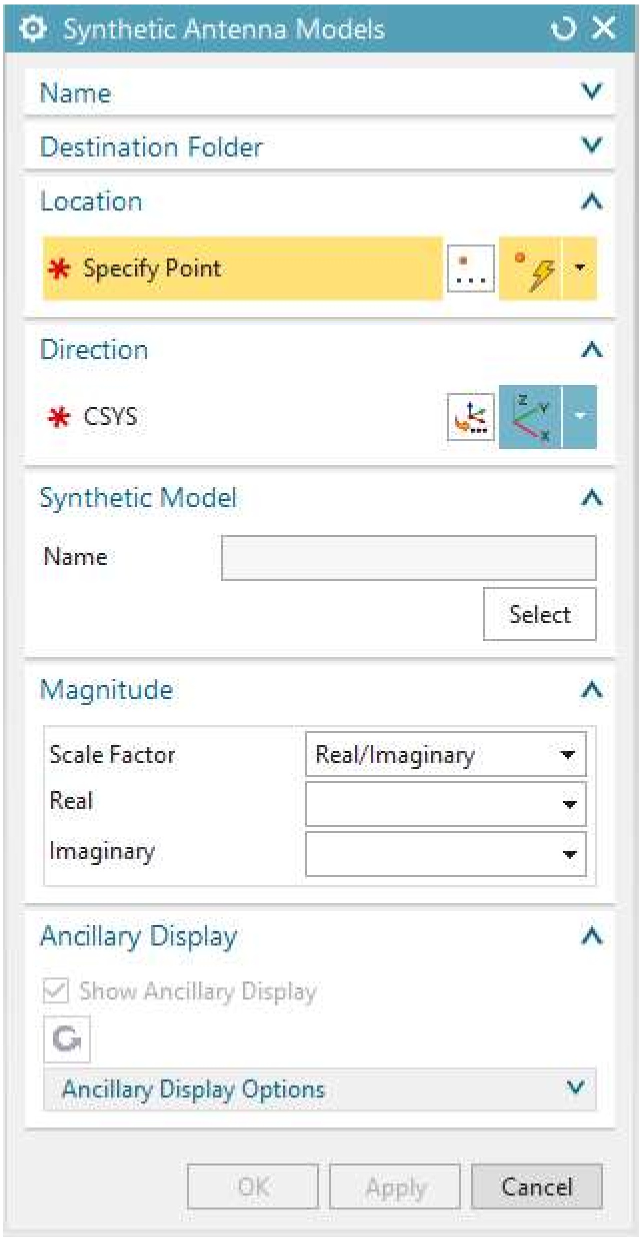
Load	
Region type list	You can choose the type of Region where the Generator Impedance is defined; available choices are: • **Voltage Generator on Element Edges** • **Voltage Generator on Element Node** • **Voltage Generator on Polygon Edge**
Name	Defines the object name. Note: these types of objects are used in *3D MoM, 2.5D MoM* and *S-PEEC* analyses only.
Description	You can enter a brief description here (optional).
Destination Folder	
Load container	Specifies the container of the load. Default is the root of Load Container.
Port	
Region selector	You can select or define a region to be assigned to the port using the **Create region tool**. The available type of region depends on the selection in the **Region type list** field.
Reverse Port Polarity	Once the port region has been selected, its positive and negative sides are automatically selected. Setting **Reverse Port Polarity** switches positive and negative sides.
Magnitude	
Voltage	Specifies how the voltage generator value is defined. Available options are: • Real/Imaginary • Magnitude/Phase
Real	Available if **Voltage** is Real/Imaginary. Specifies the real part of the generator.
Imaginary	Available if **Voltage** is Real/Imaginary. Specifies the imaginary part of the generator.
Magnitude	Available if **Voltage** is Magnitude/Phase. Specifies the magnitude of the generator.
Phase	Available if **Voltage** is Magnitude/Phase. Specifies the phase of the generator.
Generator Impedance	

Type	Available types: • **RLC**: you can model the generator impedance by a series or a parallel of a resistance, a capacitance and an inductance; • **Z**: you can model the generator impedance by means of an impedance (specifying real and imaginary part); • **Y**: you can model the generator impedance by means of an admittance (specifying real and imaginary part); • **External**: the generator impedance is loaded from an external Touchstone file (.s1p or .s2p; if .s2p is used, the second port will be considered in short-circuit).
Series/Parallel	Appears when **Type** is set to **RLC**. Specifies if the generator impedance is a series or a parallel of a resistance, a capacitance and an inductance.
Resistance	Appears when **Type** is set to **RLC**. Defines (if needed) the resistance value of the generator impedance.
Inductance	Appears when **Type** is set to **RLC**. Defines (if needed) the inductance value of the generator impedance.
Capacitance	Appears when **Type** is set to **RLC**. Defines (if needed) the capacitance value of the generator impedance.
Impedance	Appears when **Type** is set to **Z** or **Y**. Specifies how the Z or Y complex value is defined. Available options are: • Real/Imaginary • Magnitude/Phase
Real	Appears when **Type** is set to **Z** or **Y** and **Impedance** is Real/Imaginary. Specifies the real part of the generator impedance.
Imaginary	Appears when **Type** is set to **Z** or **Y** and **Impedance** is Real/Imaginary. Specifies the imaginary part of the generator impedance.
Magnitude	Appears when **Type** is set to **Z** or **Y** and **Impedance** is Magnitude/Phase.

	Specifies the magnitude of the generator impedance.
Phase	Appears when **Type** is set to **Z** or **Y** and **Impedance** is Magnitude/Phase. Specifies the phase of the generator impedance.
Touchstone file	Appears when **Type** is set to **External**. You can select the external impedance Touchstone file (.s1p or .s2p; if .s2p is used, the second port will be considered in short-circuit).

Tab. 5-19 – "Ports Excitations" GUI fields

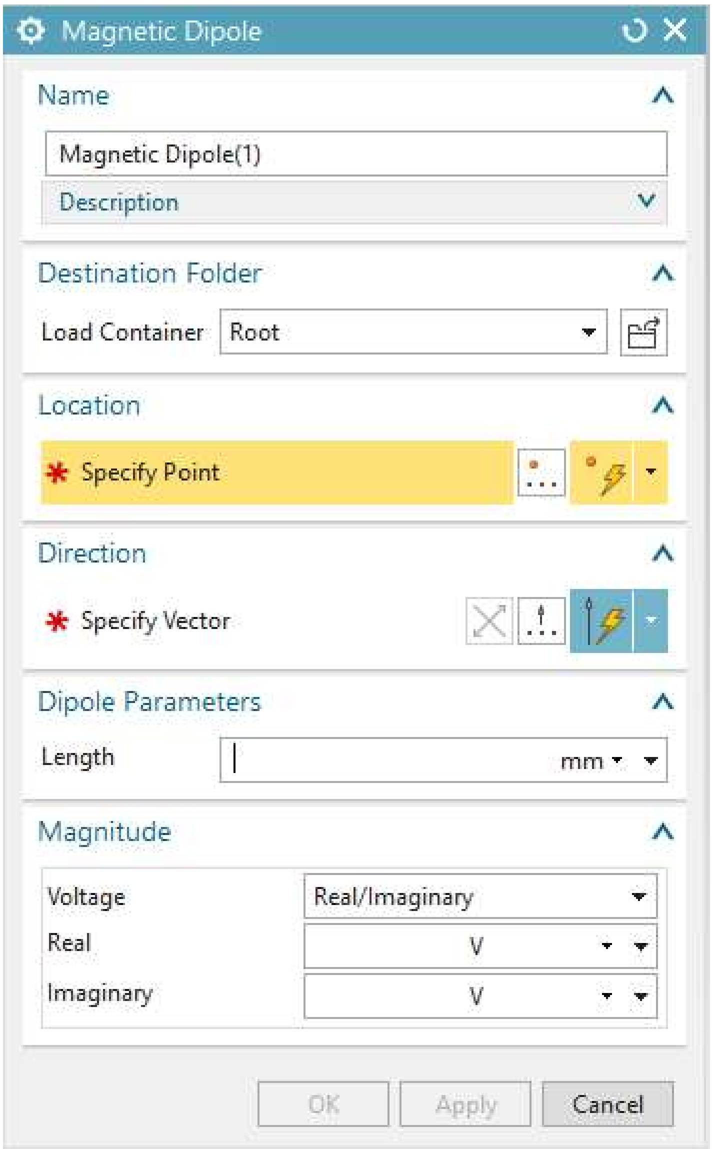
5.6.2 Synthetic Antenna Models



Name	
Name	Defines the load name.
Description	You can enter a brief description here (optional).
Destination Folder	
Load container	Specifies the container of the load. Default is the root of Load Container.
Location	
Specify Point	You can specify the antenna model installation point, using the **Point tool dialog box**.
Direction	
CSYS	You can specify the antenna orientation selecting its reference system.
Synthetic Model	
Select	Browse the Synthetic Antenna Models Library to select the desired model. The browser also you to create new models. The directory storing the antenna models library is defined as a Simcenter 3D High Frequency EM customer option.
Magnitude	
Scale Factor	Specifies how the antenna model scale factor is defined. Available options are: • Real/Imaginary • Magnitude/Phase
Real	Available if **Scale Factor** is Real/Imaginary. Specifies the real part of the Scale Factor.
Imaginary	Available if **Scale Factor** is Real/Imaginary. Specifies the imaginary part of the Scale Factor.
Magnitude	Available if **Scale Factor** is Magnitude/Phase. Specifies the magnitude of the Scale Factor.
Phase	Available if **Scale Factor** is Magnitude/Phase. Specifies the phase of the Scale Factor.

Tab. 5-20 – “Synthetic Antenna Model” GUI fields

5.6.3 Magnetic Dipole

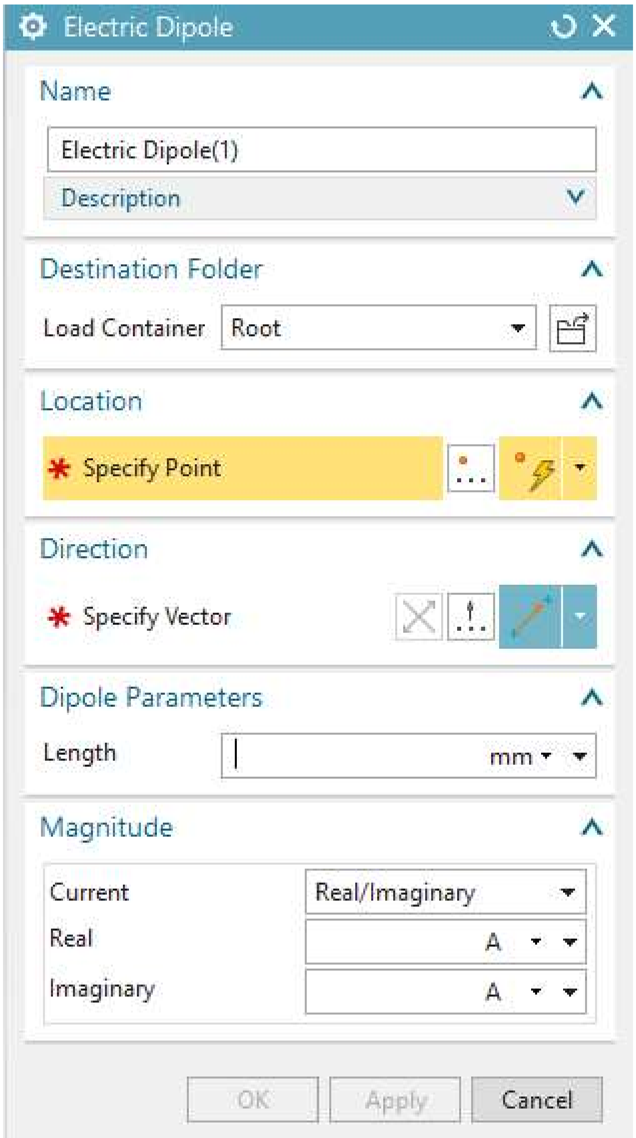


Name	
Name	Defines the load name.
Description	You can enter a brief description here (optional).
Destination Folder	
Load container	Specifies the container of the load. Default is the root of Load Container.
Location	
Specify Point	You can specify the dipole installation point, using the **Point tool dialog box**.

Direction	
Specify Vector	You can specify the dipole orientation, using the **Vector tool dialog box**.
Dipole Parameters	
Length	Specifies the dipole length.
Excitation	
Voltage	Specifies how the dipole excitation is defined. Available options are: • Real/Imaginary • Magnitude/Phase
Real	Available if **Voltage** is Real/Imaginary. Specifies the real part of the dipole excitation.
Imaginary	Available if **Voltage** is Real/Imaginary. Specifies the imaginary part of the dipole excitation.
Magnitude	Available if **Voltage** is Magnitude/Phase. Specifies the magnitude of the dipole excitation.
Phase	Available if **Voltage** is Magnitude/Phase. Specifies the phase of the dipole excitation.

Tab. 5-21 – “Magnetic Dipole” GUI fields

5.6.4 Electric Dipole

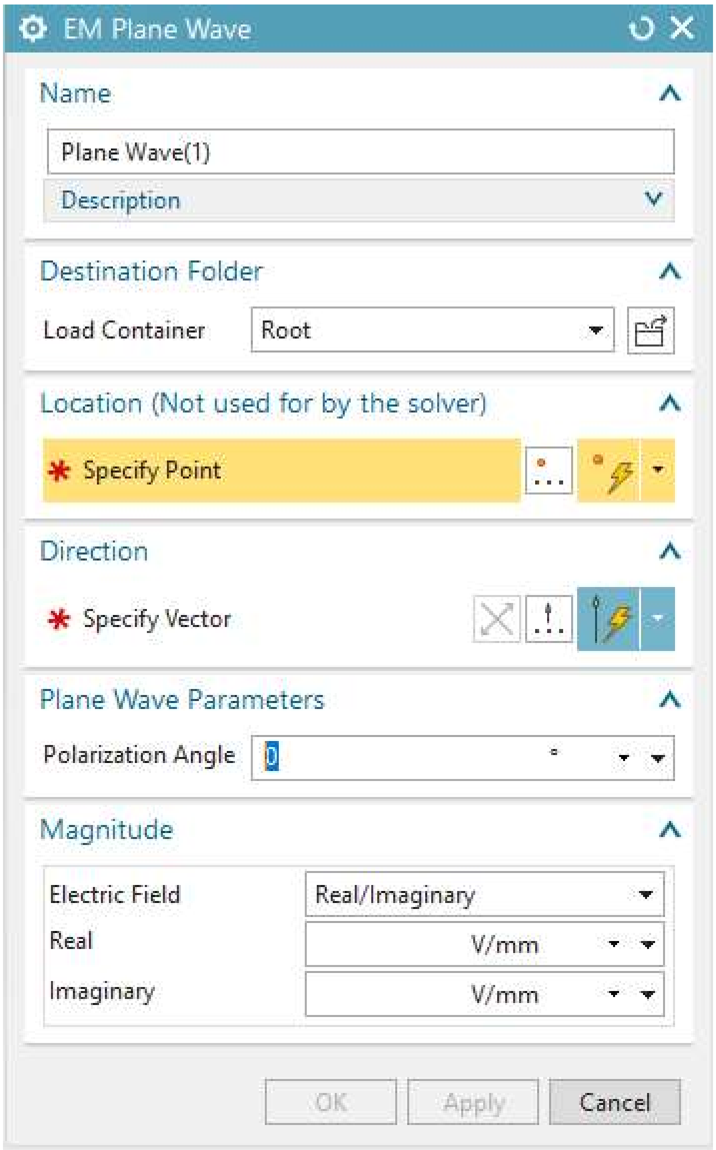


Name	
Name	Defines the load name.
Description	You can enter a brief description here (optional).
Destination Folder	
Load container	Specifies the container of the load. Default is the root of Load Container.
Location	
Specify Point	You can specify the dipole installation point, using the **Point tool dialog box**.
Direction	

Specify Vector	You can specify the dipole orientation, using the **Vector tool dialog box**.
Dipole Parameters	
Length	Specifies the dipole length.
Excitation	
Current	Specifies how the dipole excitation is defined. Available options are: • Real/Imaginary • Magnitude/Phase
Real	Available if **Current** is Real/Imaginary. Specifies the real part of the dipole excitation.
Imaginary	Available if **Current** is Real/Imaginary. Specifies the imaginary part of the dipole excitation.
Magnitude	Available if **Current** is Magnitude/Phase. Specifies the magnitude of the dipole excitation.
Phase	Available if **Current** is Magnitude/Phase. Specifies the phase of the dipole excitation.

Tab. 5-22 – “Electric Dipole” GUI fields

5.6.5 EM Plane Wave



Name	
Name	Defines the load name.
Description	You can enter a brief description here (optional).
Destination Folder	
Load container	Specifies the container of the load. Default is the root of Load Container.
Location	
Specify Point	Not used by the solver; the selected point is used to show the plane wave load symbol in the 3D visualizer.

Direction	
Specify Vector	You can specify the plane wave direction, using the **Vector tool dialog box**.
Plane Wave Parameters	
Polarization angle	Specifies the plane wave polarization angle.
Magnitude	
Electric Field	Specifies how the plane wave magnitude is defined. Available options are: • Real/Imaginary • Magnitude/Phase
Real	Available if **Electric Field** is Real/Imaginary. Specifies the real part of the plane wave magnitude.
Imaginary	Available if **Electric Field** is Real/Imaginary. Specifies the imaginary part of the plane wave magnitude.
Magnitude	Available if **Electric Field** is Magnitude/Phase. Specifies the magnitude of the plane wave magnitude.
Phase	Available if **Electric Field** is Magnitude/Phase. Specifies the phase of the plane wave magnitude.

Tab. 5-23 – "EM Plane Wave" GUI fields

5.7 3D MoM Analysis

5.7.1 Mesh type

Section 5.1 contains the description of the available mesh types for all analysis types.

5.7.2 Physical Property Tables

The following Physical Property Tables apply to *3D MoM* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Multilayer* (5.2.2)
- *Material Layer* (5.2.3)
- *Tabulated Data* (5.2.4)
- *Material Tabulated* (Hidden 5.2.5)
- *Grid* (5.2.6)
- *Surface Impedance* (5.2.7)
- *Volumetric EM Data* (5.2.8)
- *Volumetric EM Data Child* (5.2.9)

5.7.3 Modeling Objects

The following Modeling Objects apply to *3D MoM* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Frequency Scan* (5.3.1)
- *Near Field Scan Area* (5.3.2)
- *Far Field Scan Area* (5.3.3)

5.7.4 Simulation Objects

The following Simulation Objects apply to *3D MoM* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Lumped Impedance* (5.5.1)
- *Infinite Ground Plane* (5.5.2)

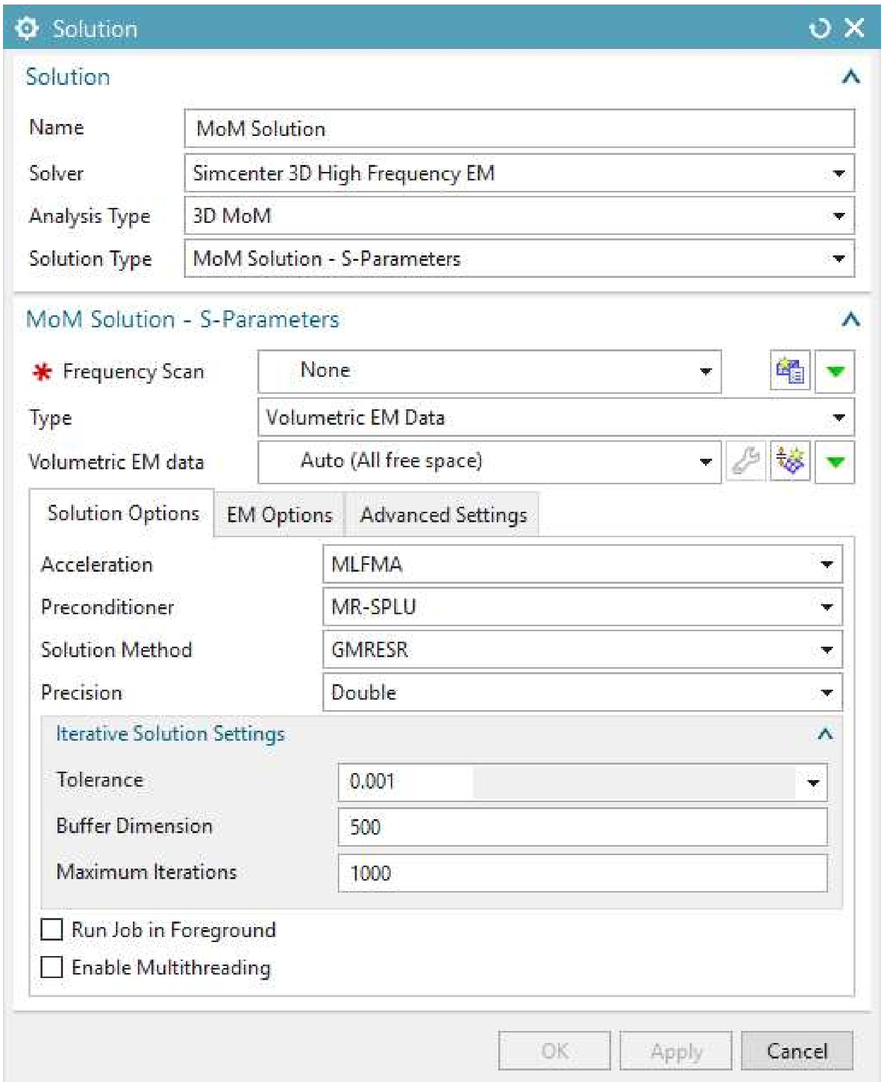
5.7.5 Loads

The following Loads apply to *3D MoM* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Port Excitations* (5.6.1)
- *Synthetic Antenna Models* (5.6.2) without *Pattern*
- *Magnetic Dipole* (5.6.3)
- *Electric Dipole* (5.6.4)
- *EM Plane Wave* (5.6.5)

5.7.6 Solutions

5.7.6.1 MoM Solutions – S-Parameters



Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *3D MoM*.
Solution Type	Specifies the solution type, in this case *MoM Solution – S-Parameters*.
MoM Solution – S-Parameters	
Frequency scan	You can select one of the available frequency scans (the analysis frequencies) or create a new one. For details see Par. 5.3.1.

<table>
<tr><td>Volumetric EM data</td><td>Defines the electromagnetic data for enclosed volumes in the model.
It is possible to:
use Auto (All free-space) option: the code assumes that all the closed surfaces in the model contain air material.
select or define a custom volumetric data: all the closed surfaces in the model are filled with the specified materials.
For details see Par. 5.2.8.</td></tr>
<tr><td colspan="2">Solution Options</td></tr>
<tr><td>Acceleration</td><td>Specifies the acceleration algorithm. Available algorithms are:
None: the standard MoM approach will be used. This approach is very RAM consuming so it is only suitable for very small models. Generic dielectrics can be modelled (PMCWHT formulation).
MLFMA, to use the Multi-Level Fast Multipole Algorithm. The suitable formulation will be chosen automatically depending on the model characteristics (4.3.1).
<u>Note</u>: if CFIE is selected in the EM Options tab, the Combined Electric Field Integral Equation (CFIE) formulation is applied. CFIE formulation is suitable for meshes of closed surface (or at least with more closed surfaces than open surfaces).</td></tr>
<tr><td>Preconditioner</td><td>You can select the preconditioner from among the ones available.
If the Acceleration algorithm is set to None, possible options are:
None: preconditioner it is not used.
MR: Multi-Resolution, selects the most appropriate available preconditioner depending on the problem under analysis.
If MLFMA Acceleration algorithm is selected, possible options are:
MR-SPLU: Multi-Resolution/Sparse LU is very fast in the preconditioner evaluation but it requires more RAM with respect to MR-ILUT.
MR-ILUT: Multi-Resolution/Incomplete LU requires less RAM with respect to MR-SPLU but it is slower in the preconditioner evaluation.
SPLU: Sparse LU, is the most expensive in terms of RAM but the most effective in terms of</td></tr>
</table>

	reducing the number of iterations to solve the problem. • **None:** no preconditioner is used.
Solution Method	If the Acceleration algorithm is set to **None**, possible options are: • **Direct**: direct MoM matrix inversion using LU factorization. This method is RAM and time consuming (respectively $\propto N^2$ and $\propto N^3$ where N is the number of unknowns) and can be applied only to small models (limited number of unknowns). If **MLFMA** Acceleration algorithm is selected, possible options are: • **BICGSTAB** (BI-Conjugate STABilized): iterative solution method. This method is suitable for well-conditioned problems (e.g. when MLFMA/CFIE acceleration algorithm can be applied). The method itself requires little RAMbut might have convergence problems also when a preconditioner is used. • **GMRESR** (Generalized Minimal RESidual): iterative solution method. It has less convergence problems (less iterations) with respect to BICGSTAB but it requires more RAM. • **FGMRES** (Flexible Generalized Minimal RESidual): iterative solution method. Not requiring a preconditioner, on average it requires less RAM than GMRESR.
Precision	Specifies the single or double precision calculation.
Iterative Solution Settings	Only available if the acceleration algorithm is **MLFMA**. You can assign control parameters of the iterative methods implemented in the 3D MoM solver. • **Tolerance**: sets the tolerance for the iterative method. The iterative method stops when the residual is lower than this value. • **Buffer Dimension**: this field is enabled only when the **GMRESR** or **FGMRES** solution method is selected. It specifies the number of previous iterations considered for the information to be used to compute the current solution. This value defines the memory workspace size. The greater this value is, the faster the convergence, but it also means larger RAM required. • **Maximum Iterations**: sets the maximum number of iterations for the iterative method. The iterative

	method stops when the number of iterations exceeds this value.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If **Enable Multithreading** option is selected, specifies the number of execution threads.
EM Options	
Integration Accuracy	Selects the number of Galerkin integral points (accuracy level). Available accuracy levels are: • Medium; • High; • Very High. Note: given a mesh with 1D Wires, a higher accuracy level means a longer computation time.
S-Parameters Normalization Impedance	Specifies the characteristic impedance of the feeding transmission line with respect to which S-parameters are normalized.
Combined Field Integral Equation (CFIE)	Only available if acceleration algorithm is **MLFMA**. Enables the Multi Level Fast Multipole Algorithm (MLFMA) based on the Combined Electric Field Integral Equation (CFIE) formulation, implemented in the MoM method.
Low Frequency Stabilization	Only available if acceleration algorithm is None. If this option is enabled a typical numerical error of the MoM approach at very low frequency (mesh length < $\lambda/106$) is removed. This option is suitable for lightning and other very low frequency problems.
Advanced Settings	
Fast Source Evaluation	If this option is enabled, an optimized algorithm for managing 2D and 3D current distribution is adopted. This reduces the computational time.
Acceleration Settings	Only available if the acceleration algorithm is **MLFMA**. Specifies the settings for the acceleration algorithm. • **Near Matrix Extension [lambda]**: size of the finest block of the multilevel spatial decomposition in terms of wavelength. The interactions relevant to the same block and to adjacent blocks are computed through a standard MoM technique and stored in the near-field sparse matrix. This parameter has a significant influence on the memory requirement. Suggested

<table>
<tr>
<td></td>
<td>
<ul style="list-style:none">
values are between 0.15 and 0.5. Default value is 0.22.

MLFMA Accuracy:

Standard: the internal error in the MLFMA expansion is below an internal threshold on an average basis. This approach assures accurate results in almost all cases.
High: the internal error in the MLFMA expansion is below an internal threshold, even in the worst case. This approach requires more RAM and computational time but could give very accurate results for extremely low field levels.

Spherical Wave Expansion: this option affects the calculation of the “far” contributions to the solution.
In normal cases, the “strong” part of the matrix solution is directly calculated and stored, while the “far” part of the solution is calculated on-the-fly during each iteration using local plane wave expansion.
Instead, using this option, a spherical wave expansions is applied: this approach requires less RAM with respect to the local plane wave expansion but it is slower.

</td>
</tr>
<tr>
<td>Preconditioner Settings</td>
<td>
Only available if the acceleration algorithm is MLFMA and preconditioner is MR-SPLU or MR-ILUT.

Specifies the settings for the preconditioner.

If the selected preconditioner is MR-SPLU, the available settings are:

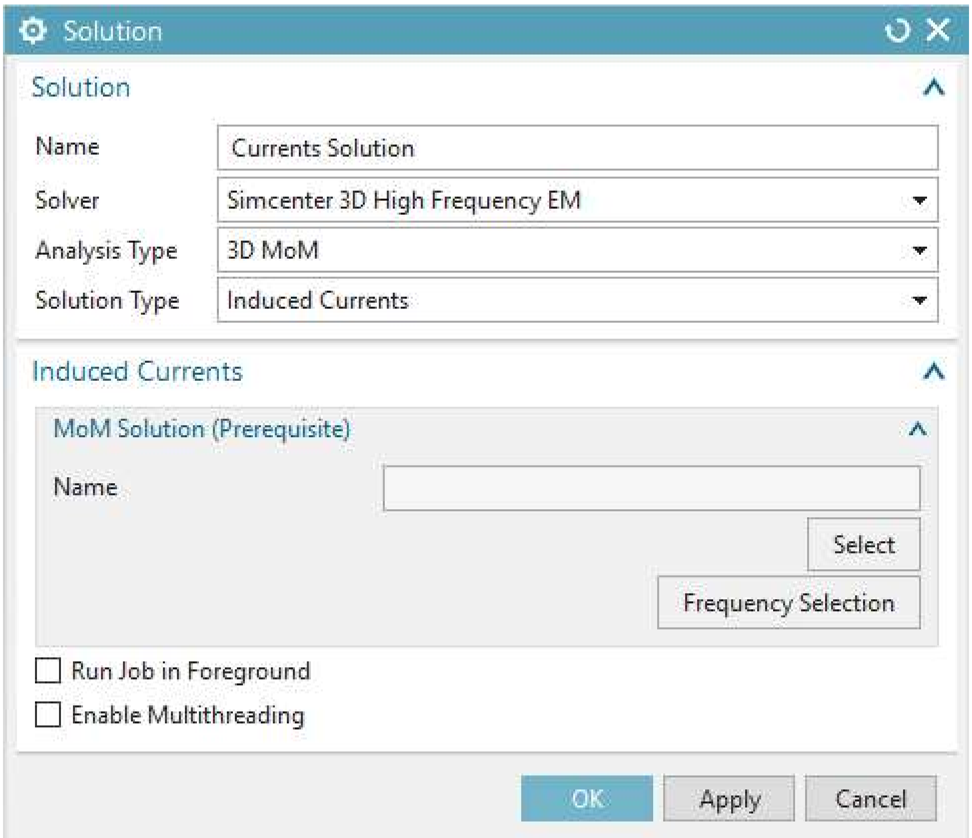
Near Matrix Clipping Threshold: clipping threshold used to sparsify the near-field sparse matrix in order to generate the input ILUT preconditioning matrix. Decreasing this parameter can improve the convergence speed of the iterative solver. Suggested values are between 1e-6 and 1e-3. Default value is 1e-3.
MR Collecting Threshold [lambda]: max group size of the Multiresolution preconditioning technique in terms of wavelength. This parameter affects the convergence speed of the iterative solver. Suggested values are between 0.05 and 0.25. Default value is 0.125.

If the selected preconditioner is MR-ILUT, the available settings are:
</td>
</tr>
</table>

	• **ILUT Matrix Filling Factor**: ratio of the dimension of the ILUT preconditioning matrix with respect to the input matrix. Increasing this parameter can improve the convergence speed of the iterative solver. Suggested values are between 2 and 6. The Default value is 4. • **Near Matrix Clipping Threshold**: see above. • **MR Collecting Threshold [lambda]**: see above.

Tab. 5-24 – "3D MoM Solutions – S-Parameters Solution" GUI fields

5.7.6.2 Induced Currents

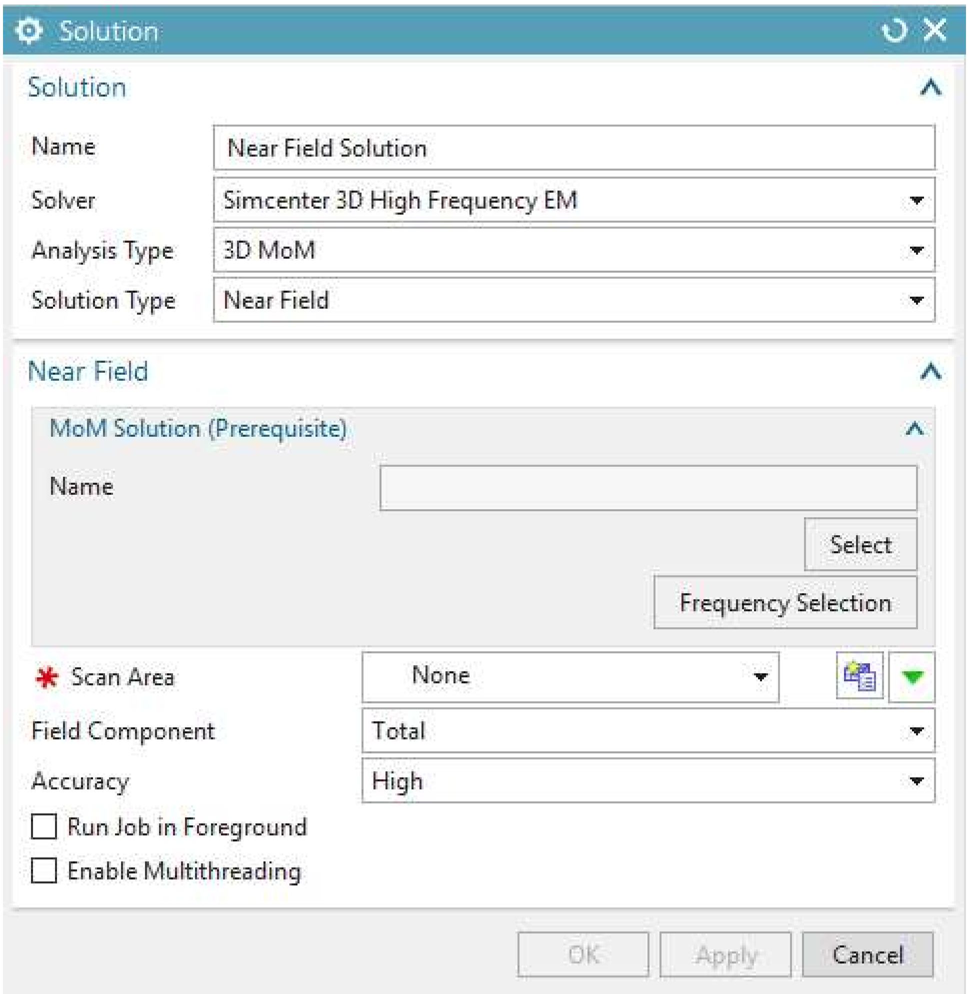


Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *3D MoM*.
Solution Type	Specifies the solution type, in this case *Induced Currents*.
Induced Currents	

MoM Solution (Prerequisite)	You can select and show a 3D MoM solution, as the prerequisite solution. • **Select**: you can select the prerequisite 3D MoM solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-25 – "3D MoM - Induced Currents Solution" GUI fields

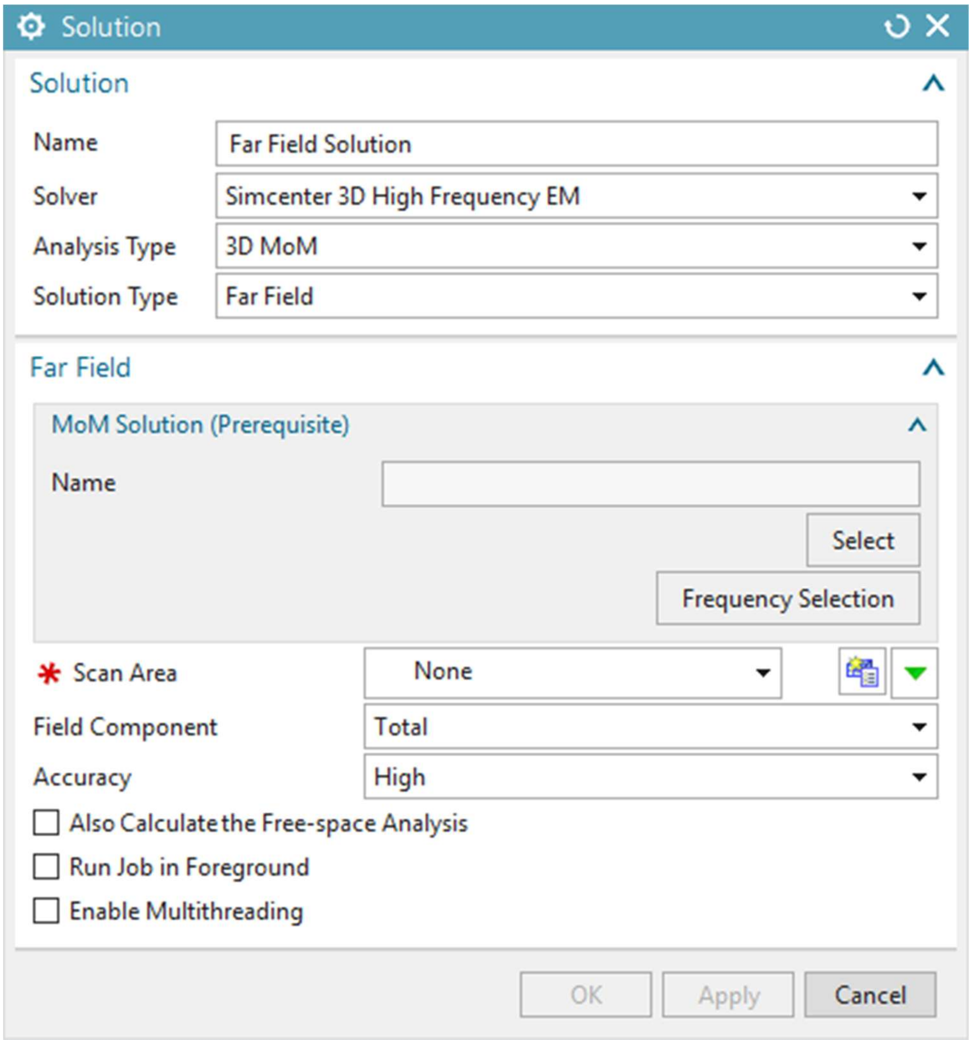
5.7.6.3 Near Field



Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *3D MoM*.
Solution Type	Specifies the solution type, in this case *Near Field*.
Near Field	
MoM Solution (Prerequisite)	You can select and show a 3D MoM solution, as the prerequisite solution. • **Select**: you can select the prerequisite 3D MoM solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Scan Area	You can select one of the available near field scans (the analysis geometrical scan) or create a new one. For details see Par.5.3.2.
Field Component	You can select the field type to be computed. Available options are: • **Total**, the total field. • **Incident**, the field radiated by the source. • **Scattered**, the field scattered from the structure.
Accuracy	You can select the accuracy level. Available options are: **Medium**, **High**, **Very High**.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-26 – “3D MoM - Near Field Solution” GUI fields

5.7.6.4 Far Field

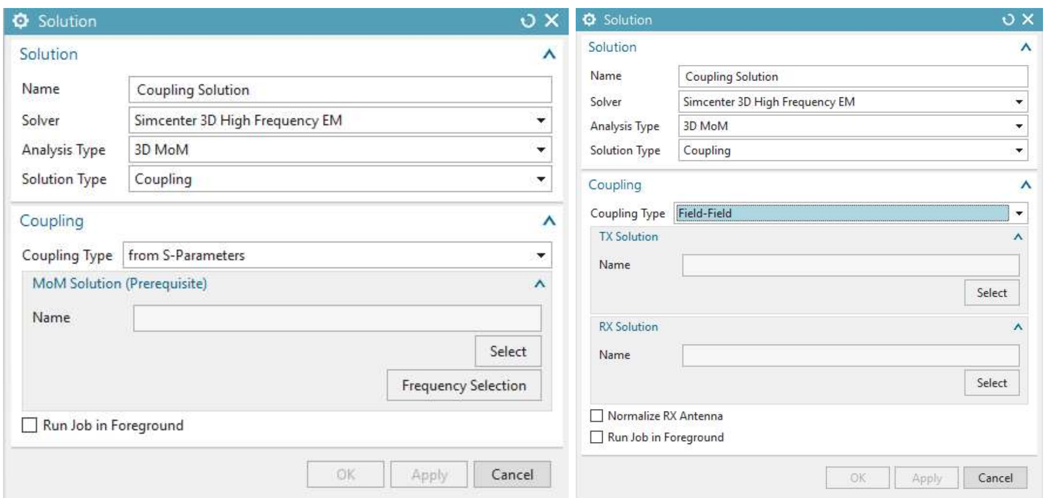


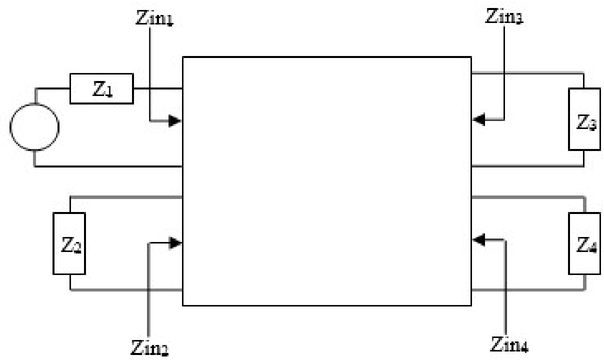
Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *3D MoM*.
Solution Type	Specifies the solution type, in this case *Far Field*.
Far Field	
MoM Solution (Prerequisite)	You can select and show a 3D MoM solution, as the prerequisite solution. • **Select**: you can select the prerequisite 3D MoM solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.

Scan Area	You can select one of the available far field scans (the analysis geometrical scan) or create a new one. For details see Par.5.3.3.
Field Component	You can select the field type to be computed. Available options are: • **Total**, the total field. • **Incident**, the field radiated by the source. • **Scattered**, the field scattered from the structure.
Accuracy	You can select the accuracy level. Available options are: **Medium**, **High**, **Very High**.
Also calculate the Free-space Analysis	If selected, the solution is run also using only the input Loads, without using the mesh model. In the post-processing phase, a comparison function between the regular run and the free-space one will be available. Note: available only for “Synthetic” (“Pattern”, “SWE”, “2D Current Distribution”, “3D Current Distribution”) and “Elementary” (“Electric” and “Magnetic” dipoles, “Plane wave”) sources, not for full-wave ports.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-27 – “3D MoM - Far Field Solution” GUI fields

5.7.6.5 Coupling

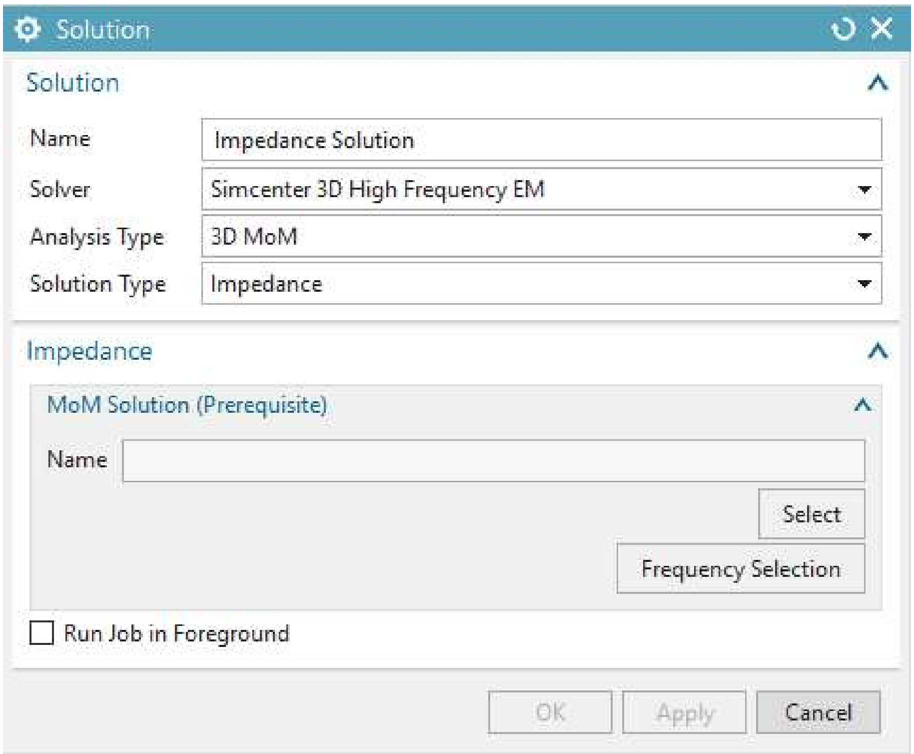


Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *3D MoM*.
Solution Type	Specifies the solution type, in this case *Coupling*.
Coupling	
Coupling type	You can select coupling type. Available options are: • **from S-Parameters**: allows the calculation of the coupling between the model ports and evaluates four different types of inter-ports coupling:  o **Active Coupling,** i.e. coupling between the source port and the victim port, taking into account the real closing impedance conditions: o **T Coupling,** Transducer Coupling, i.e. coupling taking into account the actual loading conditions for the output ports; no power reflection is taken into account at the input port. o **A Coupling,** Available Input Power Coupling, i.e. coupling under condition of maximum power transfer (Zj=Z*inj) for the output ports; impedance mismatch at the input port is taken into account. o **Maximum Coupling**, i.e. inter-ports coupling under the condition of maximum power transfer (Zj=Z*inj). These definitions come from amplifier power gain definitions, see [BD45], par. 10.5. • **Field-Field**: evaluates coupling by performing a **“reaction integral” between two conformal near fields**. The two near fields **must be evaluated on the same geometrical scan** (i.e. on the same points).

MoM Solution (Prerequisite)	Appears if the **Coupling type** is **from S-Parameters.** You can select and show a 3D MoM solution as a prerequisite solution. • **Select**: you can select the prerequisite 3D MoM solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
TX Solution	Appears if the **Coupling type** is **Field-Field.** You can select a TX solution, that is the "source" conformal near field.
RX Solution	Appears if the **Coupling type** is **Field-Field.** You can select a RX solution, that is the "victim" conformal near field.
Normalize RX Antenna	Appears if the **Coupling type** is **Field-Field.** The inter antenna coupling evaluated by means of the Field-Field integral is always normalized w.r.t. the power relevant to the Rx antenna near-field used in the calculation. The power value associated to a near-field is the input power (P_{in}) relevant to the antenna excited in the near-field evaluation. This power can differ from the radiated power (P_{rad}) for example when the antenna (or the platform) is loss affected. The coupling Field-Field integral will be normalized w.r.t.: • P_{rad} if **Normalize Rx Antenna** is checked; • P_{in} otherwise.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.

Tab. 5-28 – "3D MoM - Coupling Solution" GUI fields

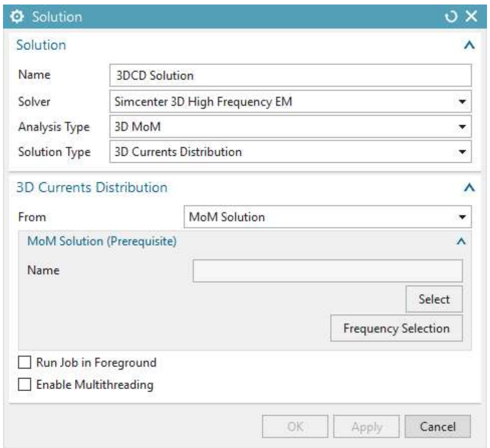
5.7.6.6 Impedance

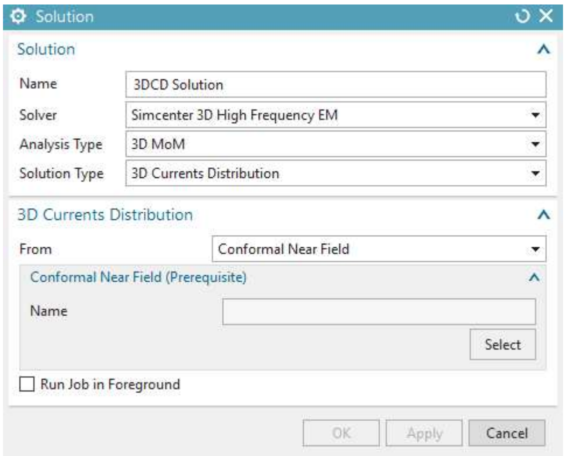


Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *3D MoM*.
Solution Type	Specifies the solution type, in this case *Impedance*.
Impedance	
MoM Solution (Prerequisite)	You can select and show a 3D MoM solution, as the prerequisite solution. • **Select**: you can select the prerequisite 3D MoM solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.

Tab. 5-29 – “3D MoM - Impedance Solution” GUI fields

5.7.6.7 3D Current Distribution





Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *3D MoM*.
Solution Type	Specifies the solution type, in this case *3D Current distribution*.
3D Current distribution	
From	Specifies the prerequisite type. Available options are: • **MoM Solution:** the model will consist of the equivalent currents corresponding to the basis functions calculated for the solution; • **Conformal Near Field:** the model will consist of the equivalent currents, $\hat{n} \times \bar{H}$ for the electric part and $\bar{E} \times \hat{n}$ for the magnetic part, placed in the same position as the mesh probe elements used to calculate the conformal near field itself.
MoM Solution (Prerequisite)	Appears if **From** is **MoM Solution.** You can select and show a MoM solution as the prerequisite solution. • **Select**: you can select the prerequisite 3D MoM solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Conformal Near Field (Prerequisite)	Appears if **From** is **Conformal Near Field.** You can select and show a Conformal Near Field solution, as the prerequisite solution.

Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	Appears if **From** is **MoM Solution.** If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-30 – “3D Current distribution Solution” GUI fields

5.8 2.5D MoM Analysis

5.8.1 Mesh type

Section 5.1 contains the description of the available mesh types for all analysis types.

5.8.2 Physical Property Tables

The following Physical Property Tables apply to 2.5*D MoM* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Multilayer* (5.2.2)
- *Material Layer* (5.2.3)
- *Tabulated Data* (5.2.4)
- *Material Tabulated* (Hidden 5.2.5)
- *Grid* (5.2.6)
- *Surface Impedance* (5.2.7)
- *Stack Layer* (5.2.10)
- *Single Layer* (5.2.11)

5.8.3 Modelling Objects

The following Modeling Objects apply to *2.5D MoM* Analyses in the *Simcenter 3D HF EM* environment:

- *Frequency Scan* (5.3.1)
- *Near Field Scan Area* (5.3.2)
- *Far Field Scan Area* (5.3.3)

5.8.4 Simulation Objects

The following Simulation Objects apply to *2.5D MoM* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Lumped Impedance* (5.5.1)

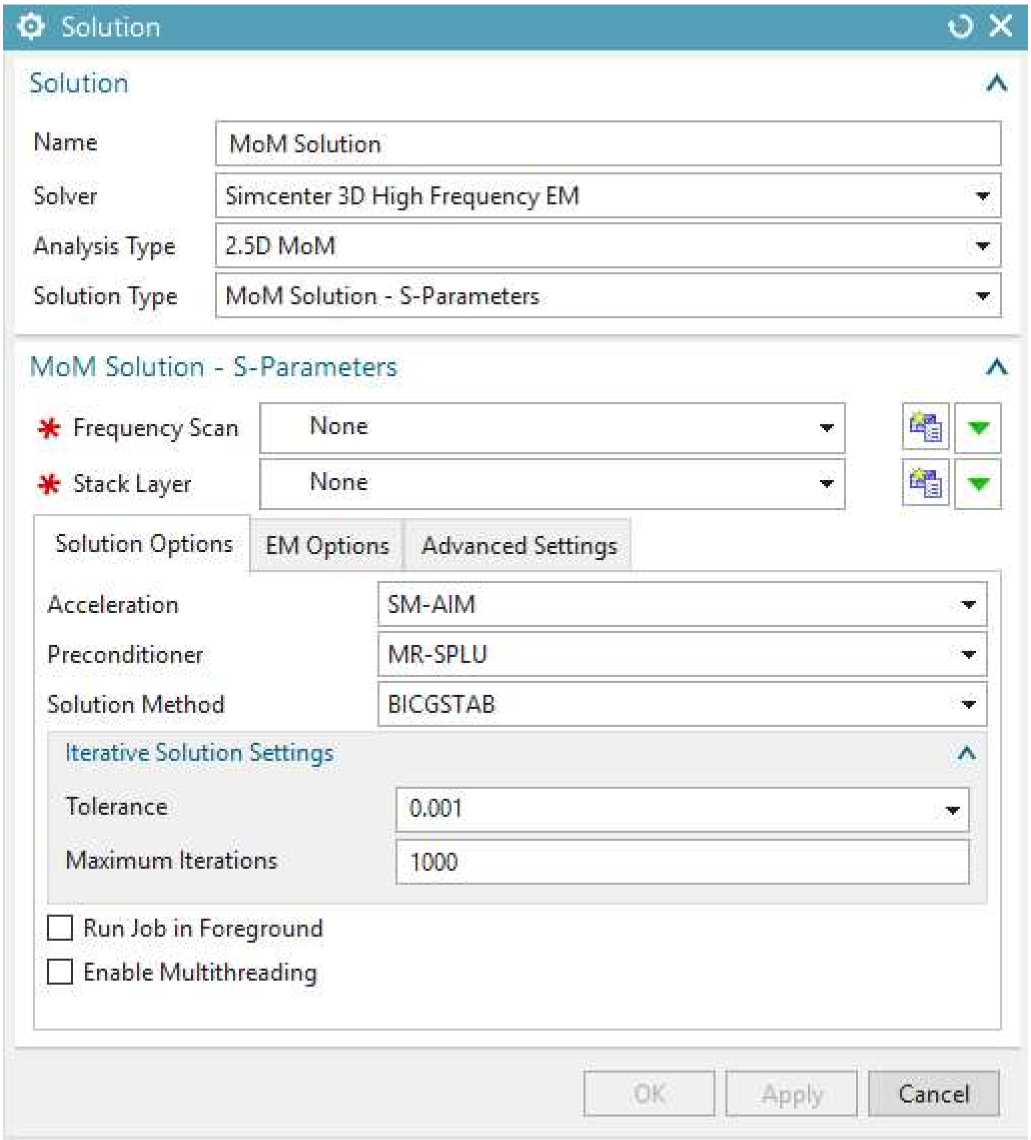
5.8.5 Loads

The following Loads apply to *2.5D MoM* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Ports Excitations* (5.6.1)

5.8.6 Solutions

5.8.6.1 MoM Solutions – S-Parameters



Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *2.5D MoM*.
Solution Type	Specifies the solution type, in this case *MoM Solution – S-Parameters*.
MoM Solution – S-Parameters	

Frequency scan	You can select one of the available frequency scans (the analysis frequencies) or create a new one. For details see Par. 5.3.1.
Stack Layer	You can select one of the available Layer Stacks or create a new one. For details see Par.5.2.10
Solution Options	
Acceleration	Specifies the acceleration algorithm. Available algorithms are: • **None**, to use the **MoM linear system** usually solved through **LU** factorization. It is characterized by a high RAM occupancy and high CPU time, therefore, is applicable to the analysis of electrically small structures. It should be noted that solving the system through the LU factorization is particularly efficient for a multiport configuration with a large number of ports since the factorization is only required to be computed once. Arbitrary shape 3D metallic objects are allowed as well as aperture on the ground planes. • **SM-AIM**, to use the **Sparse Matrix Adaptive Integral Method** algorithm. The Adaptive Integral method (AIM) is well-suited for the analysis of quasi-planar structures and appears very efficient when applied to multilayer microstrip antennas. The method is based on a modification of the classical Method of Moments, allowing a fast evaluation of the reaction integral and, when an iterative solver is used, a fast matrix-vector multiplication. Being an iterative method, the issue of convergence is of paramount importance especially in the presence of complex (multi-scale) geometries that “naturally” lead to high condition numbers. To overcome these difficulties, the Multi-resolution (MR) physics-based pre-conditioner is used to speed up the convergence of the iterative method. Its memory requirement is very limited and it is, therefore, applicable to the analysis of electrically large structures.
Preconditioner	You can select the preconditioner from among the ones available. If the Acceleration algorithm is set to **None**, possible options are:

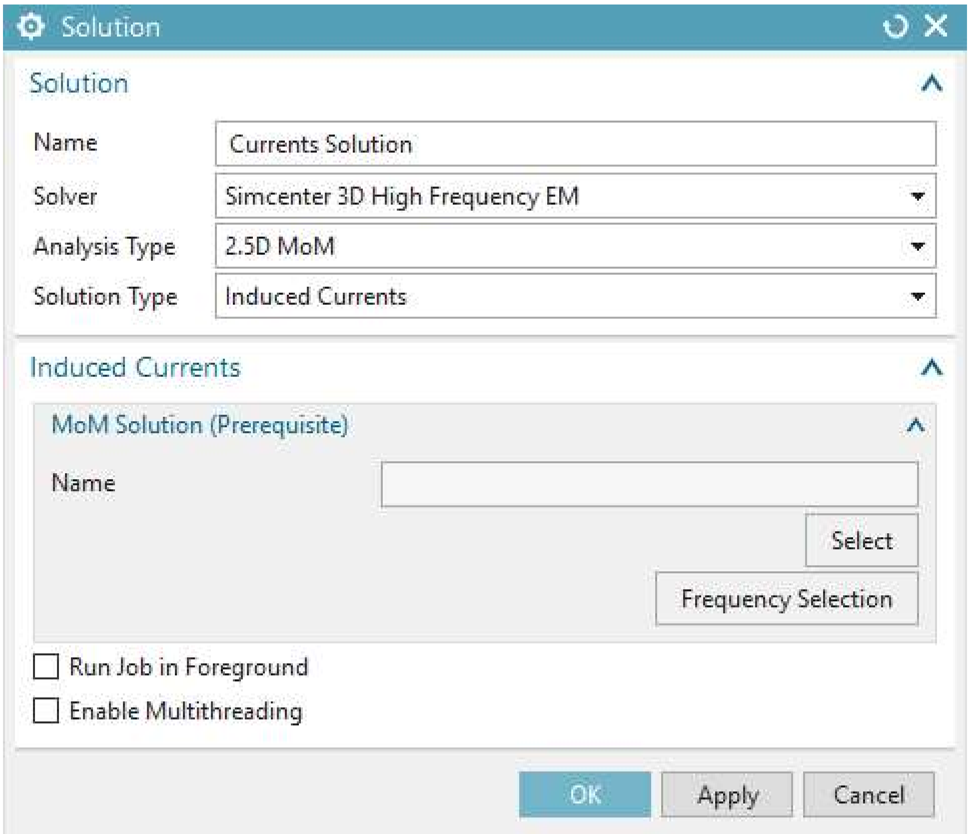
	• **None** • **MR** If **SM-AIM** Acceleration algorithm is selected, possible options are: • **MR-SPLU**: Multi-Resolution/Sparse LU, is very fast in the preconditioner evaluation but it requires more RAM with respect to MR-ILUT. • **MR-ILUT**: Multi-Resolution/Incomplete LU, requires less RAM with respect to MR-SPLU but it is slower in the preconditioner evaluation. • **SPLU**: Sparse LU, is the most expensive in terms of RAM but the most effective in terms of reducing the number of iteration to solve the problem. • **None**
Solution Method	If the Acceleration algorithm is set to **None**, possible options are: • **Direct**: direct MoM matrix inversion using LU factorization. This method is RAM and time consuming (respectively $\propto N^2$ and $\propto N^3$ where N is the number of unknowns) and can be applied only to small models (limited number of unknowns). If the **SM-AIM** Acceleration algorithm is selected, possible options are: • **BICGSTAB** (BI-Conjugate STABilized): iterative solution method. This method is suitable for well-conditioned problems (e.g. when MLFMA-CFIE acceleration algorithm can be applied). The method itself requires little RAM but might have convergence problems, even when a preconditioner is used. • **GMRESR** (GeneralizedMinimal RESidual Recursive): iterative solution method. It has less convergence problems (less iterations) with respect to BICGSTAB but it requires more RAM.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.
Iterative Solution Settings Only available if the acceleration algorithm is **SM-AIM**.	

You can assign control parameters of the iterative methods implemented in the MoM solver.	
Tolerance	Sets the tolerance for the iterative method. The iterative method stops when the residual is lower than this value.
Buffer Dimension	This field is enabled only when the **GMRESR** solution method is selected. It specifies the number of previous iterations considered for the information to be used to compute the current solution. This value defines the memory workspace size. The greater this value is, the faster the convergence, but also the larger the required RAM.
Maximum Iterations	Sets the maximum number of iterations of the iterative method. The iterative method stops when the number of iterations exceeds this value.
EM Options	
Integration Accuracy	Selects the number of Galerkin integral points (accuracy level). Available accuracy levels are: • Medium; • High; • Very High. Note: given a mesh with 1D Wires, the higher the accuracy level is the longer the computation time is.
S-Parameters Normalization Impedance	Specifies the characteristic impedance of the feeding transmission line with respect to which S-parameters are normalized.
Impenetrable Sheet Formulation	Appears if the **Acceleration** is **None.** It selects the **"Impedance Boundary Conditions"** formulation. Un-checking this option selects the approximate **RBC**, **"Resistive Boundary Condition"** formulation. The RBC formulation can be used if all materials in the model satisfy the following condition: Zs << 377Ω (good conductors)
Advanced Settings	
Acceleration Settings	Only available if the acceleration algorithm is **SM-AIM**. Specifies the settings for the acceleration algorithm. • **Grid Space**: if *checked*, enables a field to set the sampling rate of the FFT. This parameter depends on the mesh dimensions and on the wavelength λ. Suggested values range for λ/20 to λ/10. If *not checked* the parameter is automatically set by the system. • **Near Matrix Extension [lambda]**: if *checked*, enables a field to define the size of the finest block

	of the multilevel spatial decomposition in terms of wavelength. The interactions relevant to the same block and to adjacent blocks are computed through a standard MoM technique and stored in the near-field sparse matrix. This parameter has a significant influence on the memory requirement. Suggested values are between 0.15 and 0.5. Default value is 0.5. • **Multipole Order**: sets the number of poles for the multi-pole expansion of the standard MoM basis function. The default valued is 3 and is an optimized one. • **Tolerance AIM**: sets the accuracy of the Far Field interactions. The default valued is 0.005 and is an optimized one.
Preconditioner Settings	Only available if the acceleration algorithm is **SM-AIM** and the preconditioner is **MR-SPLU** or **MR-ILUT**. Specifies the settings for the preconditioner. If the selected preconditioner is **MR-SPLU**, the available settings are: • **Near Matrix Clipping Threshold**: clipping threshold used to sparsify the near-field sparse matrix in order to generate the input ILUT preconditioning matrix. Decreasing this parameter can improve the convergence speed of the iterative solver. Suggested values are between 1e-6 and 1e-3. Default value is 1e-3. • **MR Collecting Threshold [lambda]**: max group size of the Multiresolution preconditioning technique in terms of wavelength. This parameter affects the convergence speed of the iterative solver. Suggested values are between 0.05 and 0.25. Default value is 0.125. If the selected preconditioner is **MR-ILUT**, the available settings are: • **ILUT Matrix Filling Factor**: ratio of the dimension of the ILUT preconditioning matrix with respect to the input matrix. Increasing this parameter can improve the convergence speed of the iterative solver. Suggested values are between 2 and 6. Default value is 4. • **Near Matrix Clipping Threshold**: see above. • **MR Collecting Threshold [lambda]**: see above.

Tab. 5-31 – “2.5D MoM Solutions – S-Parameters Solution” GUI fields

5.8.6.2 Induced Currents

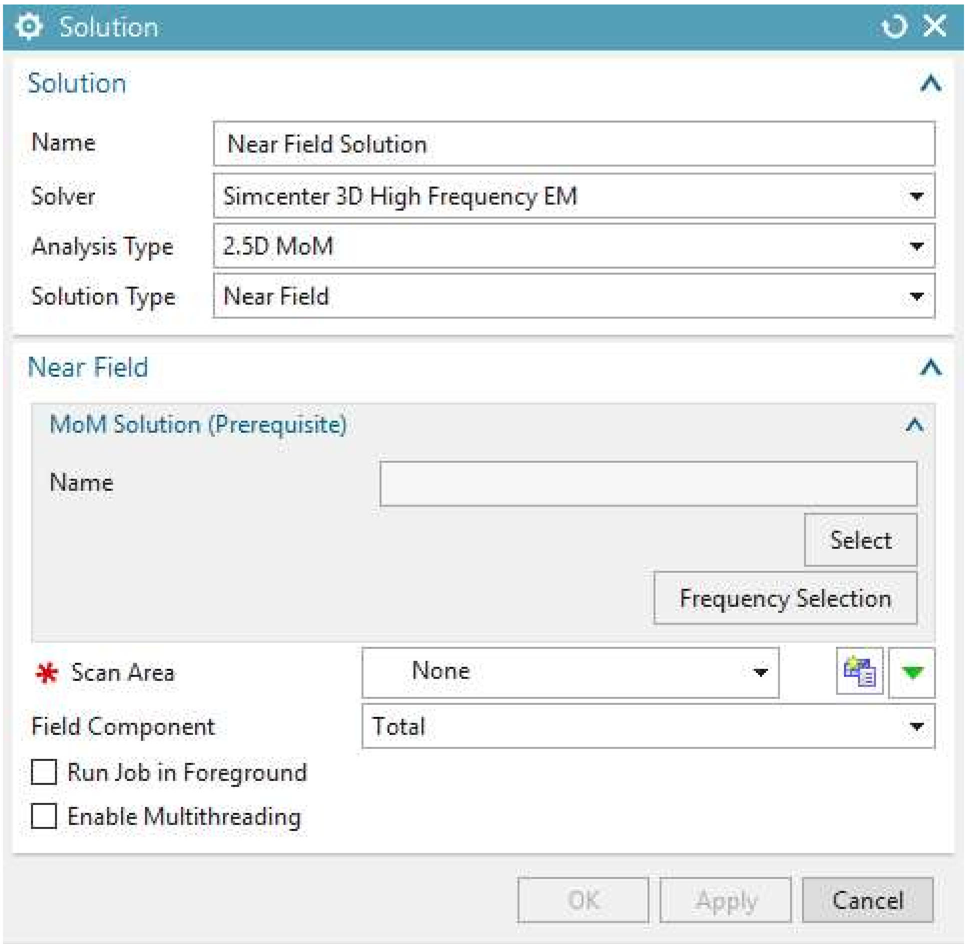


Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *2.5D MoM*.
Solution Type	Specifies the solution type, in this case *Induced Currents*.
Induced Currents	
MoM Solution (Prerequisite)	You can select and show a 2.5D MoM solution, as the prerequisite solution. • **Select**: you can select the prerequisite 2.5D MoM solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.

Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-32 – “2.5D MoM - Induced Currents Solution” GUI fields

5.8.6.3 Near Field

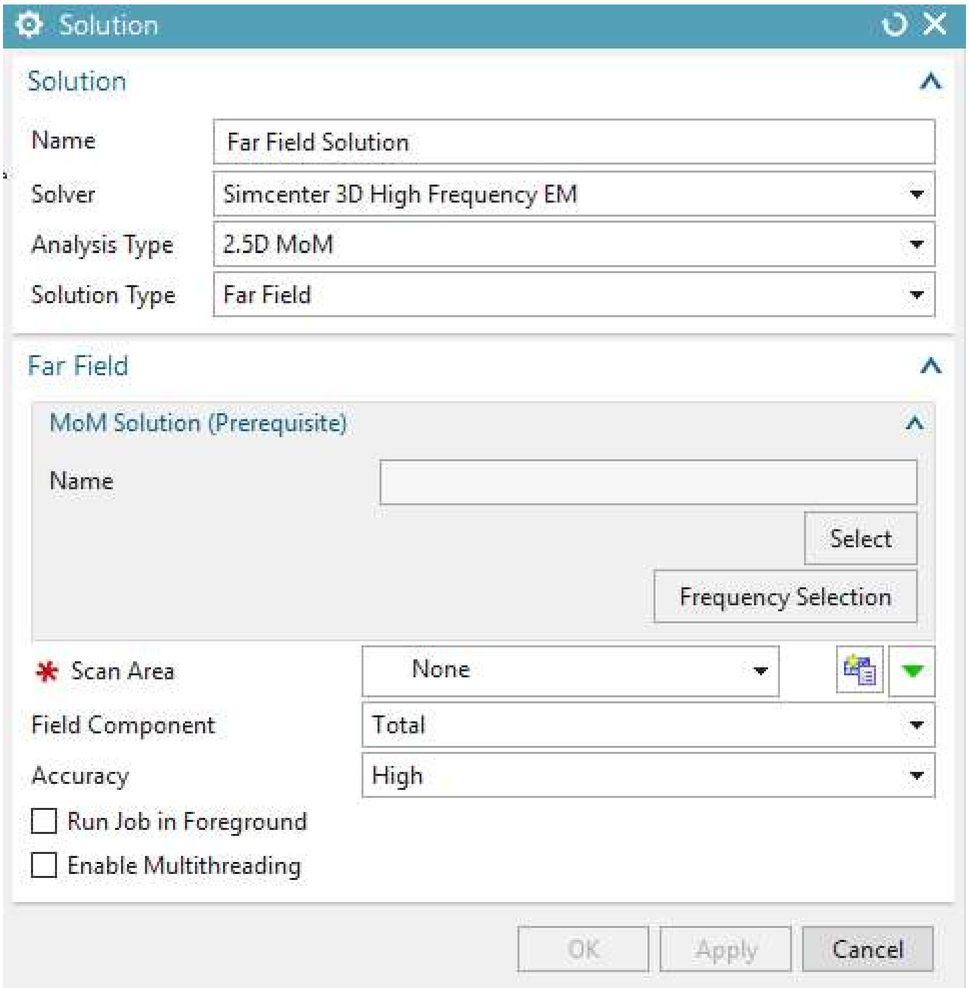


Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *2.5D MoM*.
Solution Type	Specifies the solution type, in this case *Near Field*.
Near Field	
MoM Solution (Prerequisite)	You can select and show a 2.5D MoM solution, as the prerequisite solution. • **Select**: you can select the prerequisite 2.5D MoM solution.

	• **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Scan Area	You can select one of the available near field scans (the analysis geometrical scan) or create a new one. For details see Par.5.3.2.
Field Component	You can select the field type to be computed. Available options are: • **Total**, the total field. • **Incident**, the field radiated by the source. • **Scattered**, the field scattered from the structure.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-33 – “2.5D MoM - Near Field Solution” GUI fields

5.8.6.4 Far Field

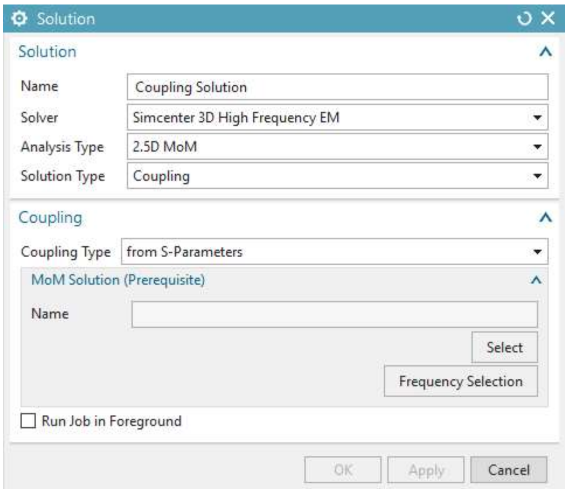


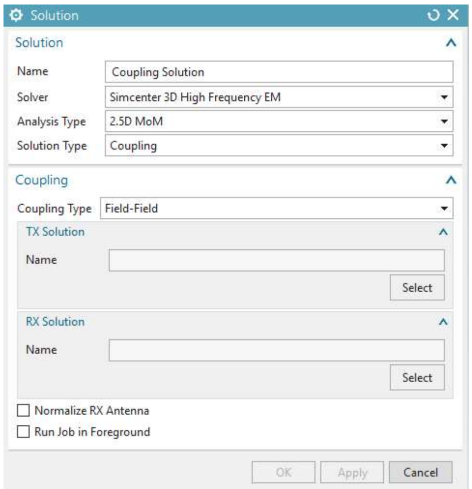
Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *2.5D MoM*.
Solution Type	Specifies the solution type, in this case *Far Field*.
Far Field	
MoM Solution (Prerequisite)	You can select and show a 2.5D MoM solution, as the prerequisite solution. • **Select**: you can select the prerequisite 2.5D MoM solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.

Scan Area	You can select one of the available far field scans (the analysis geometrical scan) or create a new one. For details see Par.5.3.3.
Field Component	You can select the field type to be computed. Available options are: • **Total**, the total field. • **Incident**, the field radiated by the source. • **Scattered**, the field scattered from the structure.
Accuracy	You can select the accuracy level. Available options are: **Medium**, **High**, **Very High**.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-34 – "2.5D MoM - Far Field Solution" GUI fields

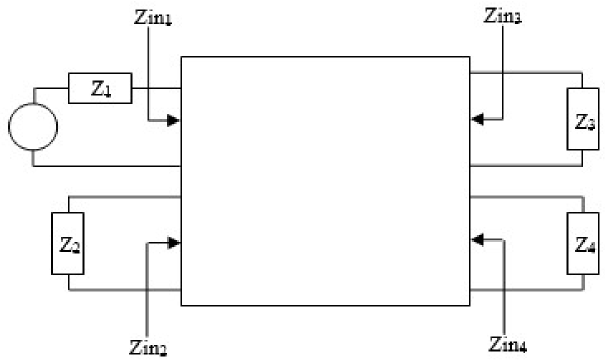
5.8.6.5 Coupling





Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *2.5D MoM*.
Solution Type	Specifies the solution type, in this case *Coupling*.
Coupling	
Coupling type	You can select coupling type.

<table>
<tr><td></td><td>Available options are:

from S-Parameters: allows the calculation of the coupling between the model ports and evaluates four different types of inter-ports coupling:


Active Coupling, i.e. coupling between the source port and the victim port, taking into account the real closing impedance conditions:
T Coupling, Transducer Coupling, i.e. coupling taking into account the actual loading conditions for the output ports; no power reflection is taken into account at the input port.
A Coupling, Available Input Power Coupling, i.e. coupling under condition of maximum power transfer (Zj=Z*inj) for the output ports; impedance mismatch at the input port is taken into account.
Maximum Coupling, i.e. inter-ports coupling under the condition of maximum power transfer (Zj=Z*inj).

These definitions come from amplifier power gain definitions, see [BD45], par. 10.5.
Field-Field: evaluates coupling by performing a “reaction integral” between two conformal near fields. The two near fields must be evaluated on the same geometrical scan (i.e. on the same points).
</td></tr>
<tr><td>MoM Solution (Prerequisite)</td><td>Appears if the Coupling type is from S-Parameters.

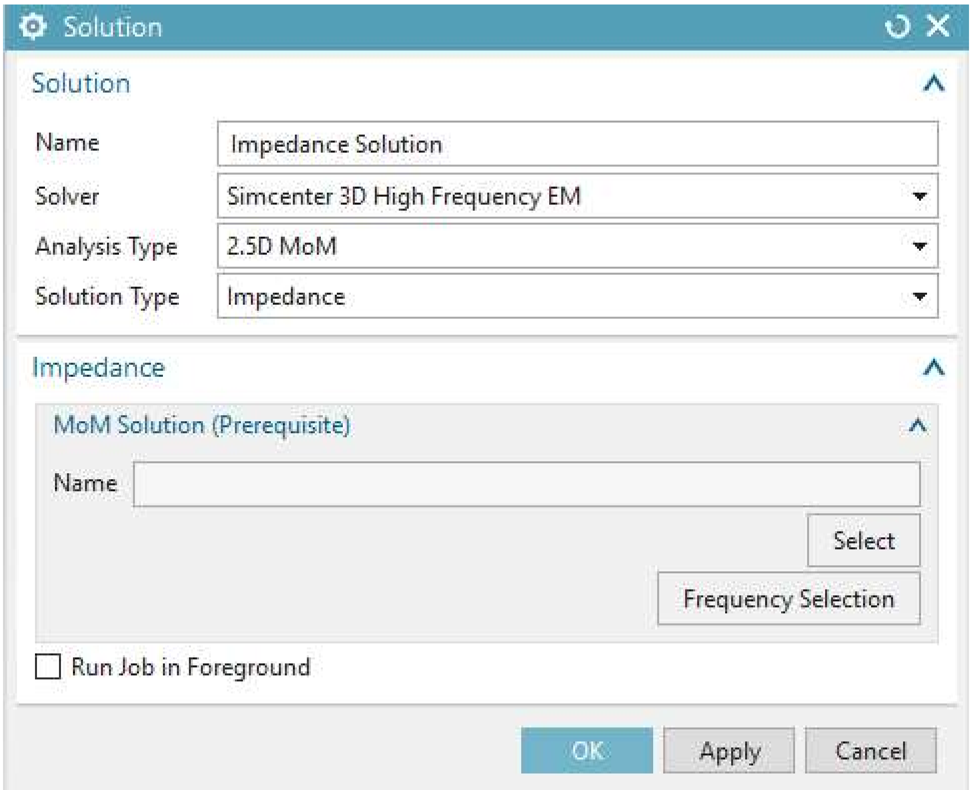
You can select and show a 2.5D MoM solution, as the prerequisite solution.

Select: you can select the prerequisite 2.5D MoM solution.
Frequency selection: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
</td></tr>
</table>

TX Solution	Appears if the **Coupling type** is **Field-Field.** You can select a TX solution, that is the "source" conformal near field.
RX Solution	Appears if the **Coupling type** is **Field-Field.** You can select a RX solution, that is the "victim" conformal near field.
Normalize RX Antenna	Appears if the **Coupling type** is **Field-Field.** The inter antenna coupling evaluated by means of the Field-Field integral is always normalized w.r.t. the power relevant to the Rx antenna near-field used in the calculation. The power value associated to a near-field is the input power (P_{in}) relevant to the antenna excited in the near-field evaluation. This power can differ from the radiated power (P_{rad}) for example when the antenna (or the platform) is loss affected. The coupling Field-Field integral will be normalized w.r.t.: • P_{rad} if the **Normalize Rx Antenna** is checked; • P_{in} otherwise.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.

Tab. 5-35 – "2.5D MoM - Coupling Solution" GUI fields

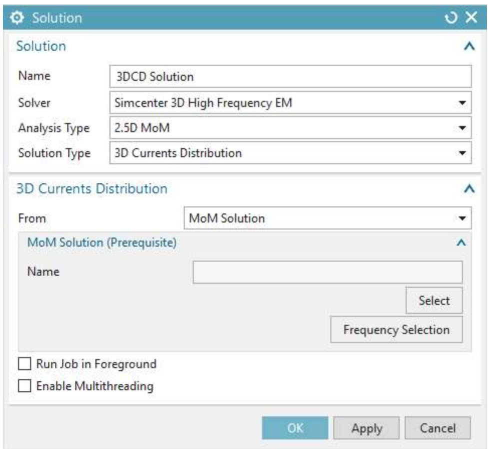
5.8.6.6 Impedance

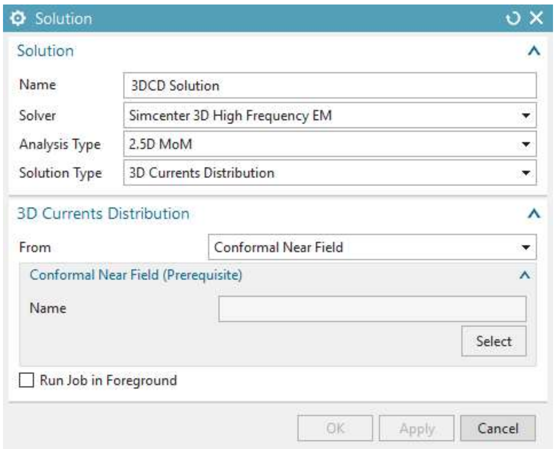


Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *2.5D MoM*.
Solution Type	Specifies the solution type, in this case *Impedance*.
Impedance	
MoM Solution (Prerequisite)	You can select and show a 2.5D MoM solution, as the prerequisite solution. • **Select**: you can select the prerequisite 2.5D MoM solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.

Tab. 5-36 – "2.5D MoM - Impedance Solution" GUI fields

5.8.6.7 3D Current Distribution





Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case 2.5*D MoM*.
Solution Type	Specifies the solution type, in this case *3D Current distribution*.
3D Current distribution	
From	Specifies the prerequisite type. Available options are: • **MoM Solution:** the model will consist of the equivalent currents corresponding to the basis functions calculated for the solution; • **Conformal Near Field:** the model will consist of the equivalent currents, $\hat{n} \times \bar{H}$ for the electric part and $\bar{E} \times \hat{n}$ for the magnetic part, placed in the same position as the mesh probe elements used to calculate the conformal near field itself.
MoM Solution (Prerequisite)	Appears if **From** is **MoM Solution.** You can select and show a 2.5D MoM solution, as the prerequisite solution. • **Select**: you can select the prerequisite 2.5D MoM solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Conformal Near Field (Prerequisite)	Appears if **From** is **Conformal Near Field.** You can select and show a Conformal Near Field solution, as the prerequisite solution.

Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	Appears if **From** is **MoM Solution.** If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-37 – "3D Current distribution Solution" GUI fields

5.9 S-PEEC Analysis

5.9.1 Mesh type

Section 5.1 contains the description of the available mesh types for all analysis types.

5.9.2 Physical Property Tables

The following Physical Property Tables apply to *S-PEEC* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Multilayer* (5.2.2)
- *Material Layer* (5.2.3)
- *Tabulated Data* (5.2.4)
- *Material Tabulated* (Hidden 5.2.5)
- *Grid* (5.2.6)
- *Surface Impedance* (5.2.7)

5.9.3 Modelling Objects

The following Modeling Objects apply to *S-PEEC* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Frequency Scan* (5.3.1)
- *Near Field Scan Area* (5.3.2)
- *Far Field Scan Area* (5.3.3)

5.9.4 Simulation Objects

The following Simulation Objects apply to *S-PEEC* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Lumped Impedance* (5.5.1)
- *Infinite Ground Plane* (5.5.2)

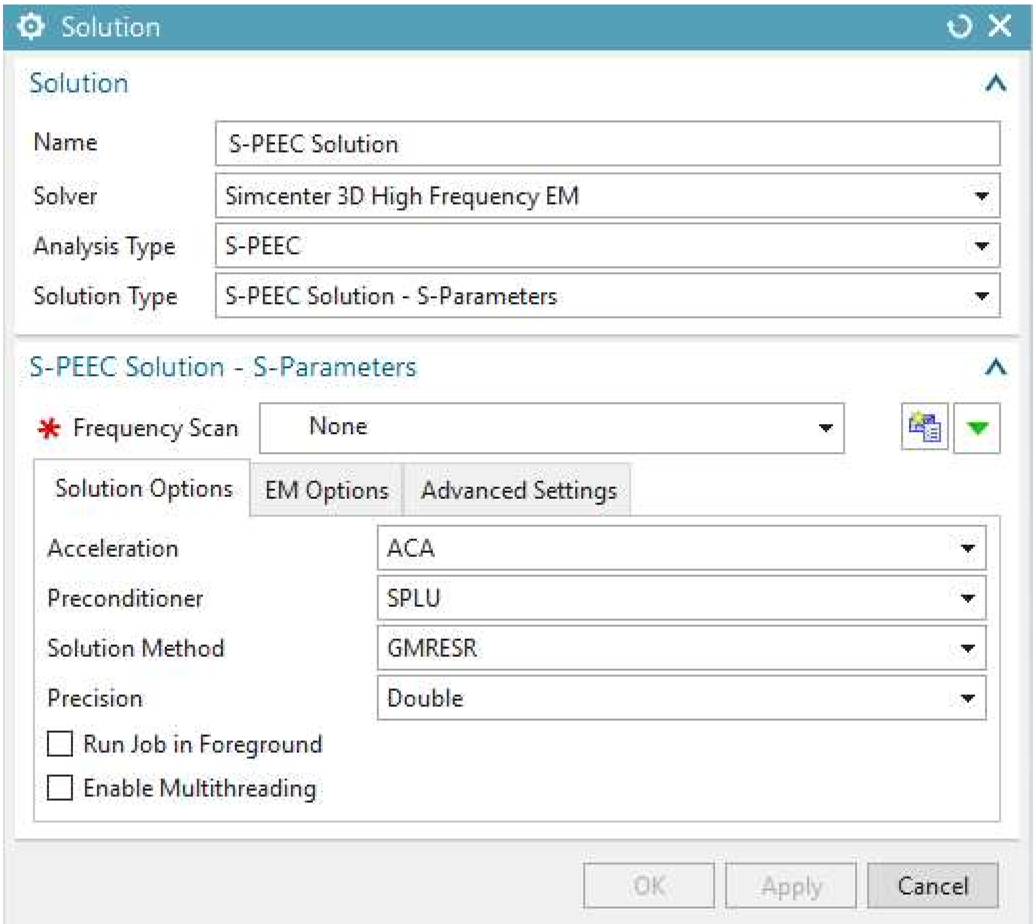
5.9.5 Loads

The following Loads apply to *S-PEEC* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Ports Excitations* (5.6.1)
- *Synthetic Antenna Models* (5.6.2)
- *Magnetic Dipole* (5.6.3)
- *Electric Dipole* (5.6.4)
- *EM Plane Wave* (5.6.5)

5.9.6 Solutions

5.9.6.1 S-PEEC Solutions – S-Parameters



Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *S-PEEC*.
Solution Type	Specifies the solution type, in this case *S-PEEC Solution – S-Parameters*.
S-PEEC Solution – S-Parameters	
Frequency scan	You can select one of the available frequency scans (the analysis frequencies) or create a new one. For details see Par. 5.3.1.
Solution Options	
Acceleration	Specifies the acceleration algorithm. Available algorithms are:

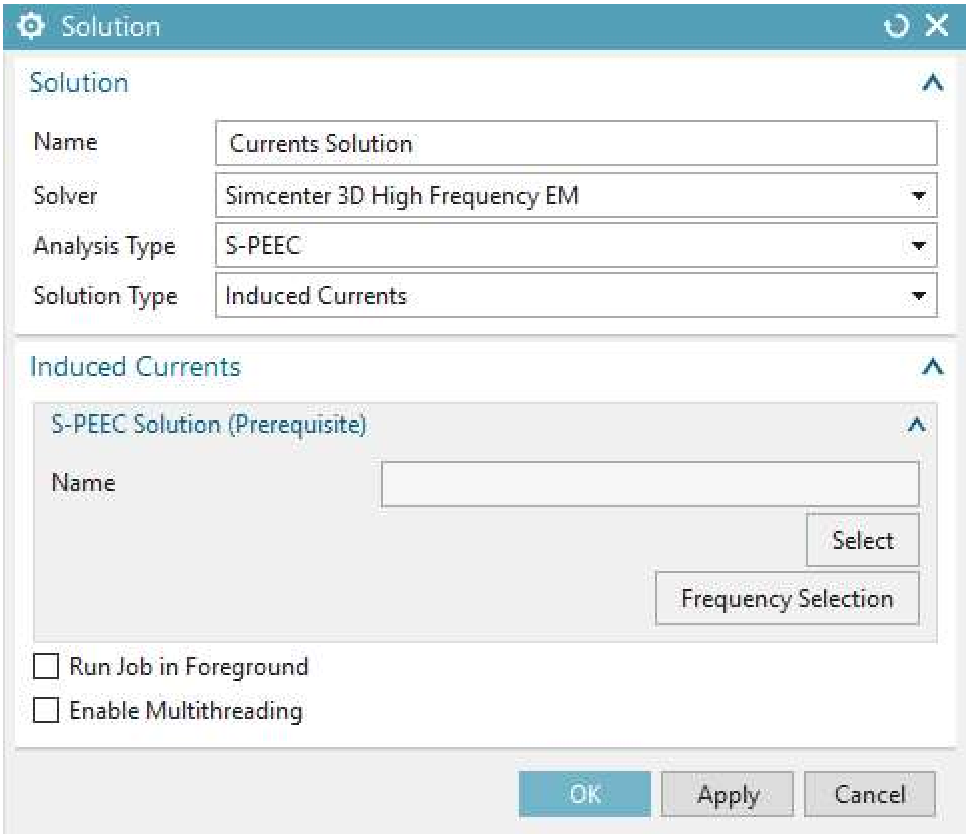
	• **None**, activates the standard S-PEEC. This approach is very RAM consuming so it is suitable for small models. • **ACA**, selects the Adaptive Cross Approximation (ACA) algorithm implemented in the S-PEEC method. This approach is less RAM and time consuming than the standard one but its convenience in terms of time decreases with increasing number of terminals/ports.
Preconditioner	You can select the preconditioner among the ones available. If the Acceleration algorithm is set to None, possible options are: • **None** If the **ACA** Acceleration algorithm is selected, possible options are: • **SPLU**: Sparse LU, is the most expensive in terms of RAM but the most effective in terms of reducing the number of iterations to solve the problem. • **None.**
Solution Method	If the Acceleration algorithm is set to None, possible options are: • **Direct**: direct MoM matrix inversion using LU factorization. This method is RAM and time consuming (respectively $\propto N^2$ and $\propto N^3$ where N is the number of unknowns) and can be applied only to small models (limited number of unknowns). • **BICGSTAB** (BI-Conjugate STABilized): iterative solution method. This method is suitable for well-conditioned problems (e.g. when MLFMA/CFIE acceleration algorithm can be applied). The method itself requires little RAM, but might have convergence problems even when a preconditioner is used. • **GMRESR** (GeneralizedMinimal RESidual Recursive): iterative solution method. It has less convergence problems (less iterations) with respect to BICGSTAB but it requires more RAM. If the **ACA** Acceleration algorithm is selected, possible options are: • **BICGSTAB**: see above. • **GMRESR**: see above. • **FGMRES** (Flexible GeneralizedMinimal RESidual): iterative solution method. This method is less RAM demanding with respect to

	GMRESR since it seldom requires the use of a preconditioner.
Precision	Specifies the single or double precision calculation.
Iterative Solution Settings	Only available if the acceleration algorithm is **ACA**. You can assign control parameters of the iterative methods implemented in the S-PEEC solver. • **Tolerance**: sets the tolerance for the iterative method. The iterative method stops when the residual is lower than this value. • **Buffer Dimension**: This field is enabled only when **GMRESR** or **FGMRES** solution method is selected. It specifies the number of previous iterations considered for the information to be used to compute the current solution. This value defines the memory workspace size. The greater this value is, the faster the convergence, but the required RAM is also greater. • **Maximum Iterations**: sets the maximum number of iterations of the iterative method. The iterative method stops when the number of iterations exceeds this value.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
the Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.
EM Options	
Integration Accuracy	Selects the number of Galerkin integral points (accuracy level). Available accuracy levels are: • Medium; • High; • Very High. Note: given a mesh with 1D Wires, the higher the accuracy level is, the longer the computation time is.
S-Parameters Normalization Impedance	Specifies the characteristic impedance of the feeding transmission line with respect to which S-parameters are normalized.
Advanced Settings	
Fast Source Evaluation	If this option is enabled, an optimized algorithm for sources modelled as equivalent Huygens sources is activated. This reduces the computational time.

Acceleration Settings	Only available if the acceleration algorithm is **ACA**. Specifies the settings for the acceleration algorithm. • **Near Matrix Width**: defines the block dimension used in the ACA approximation. The ACA technique works by subdividing the structure under analysis into blocks, and by compressing the S-PEEC interaction matrix between these blocks. In order to be effective, blocks should include tens of basis functions (i.e. tens of meshing elements). • **ACA Factorization Threshold**: defines the approximation threshold for the ACA algorithm. $1e^{-4}$ to $1e^{-6}$ are suggested values.

Tab. 5-38 – "S-PEEC Solutions – S-Parameters Solution" GUI fields

5.9.6.2 Induced Currents

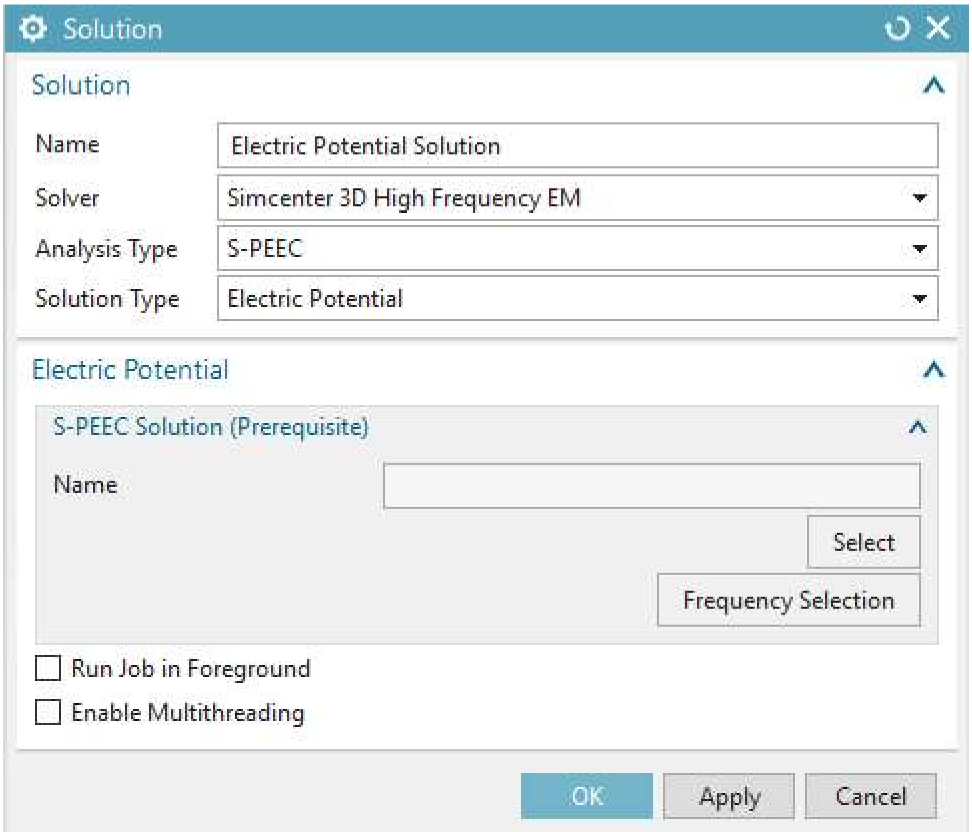


Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *S-PEEC*.

Solution Type	Specifies the solution type, in this case *Induced Currents*.
Induced Currents	
S-PEEC Solution (Prerequisite)	You can select and show a S-PEEC solution, as the prerequisite solution. • **Select**: you can select the prerequisite S-PEEC solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-39 – "S-PEEC - Induced Currents Solution" GUI fields

5.9.6.3 Electric Potential

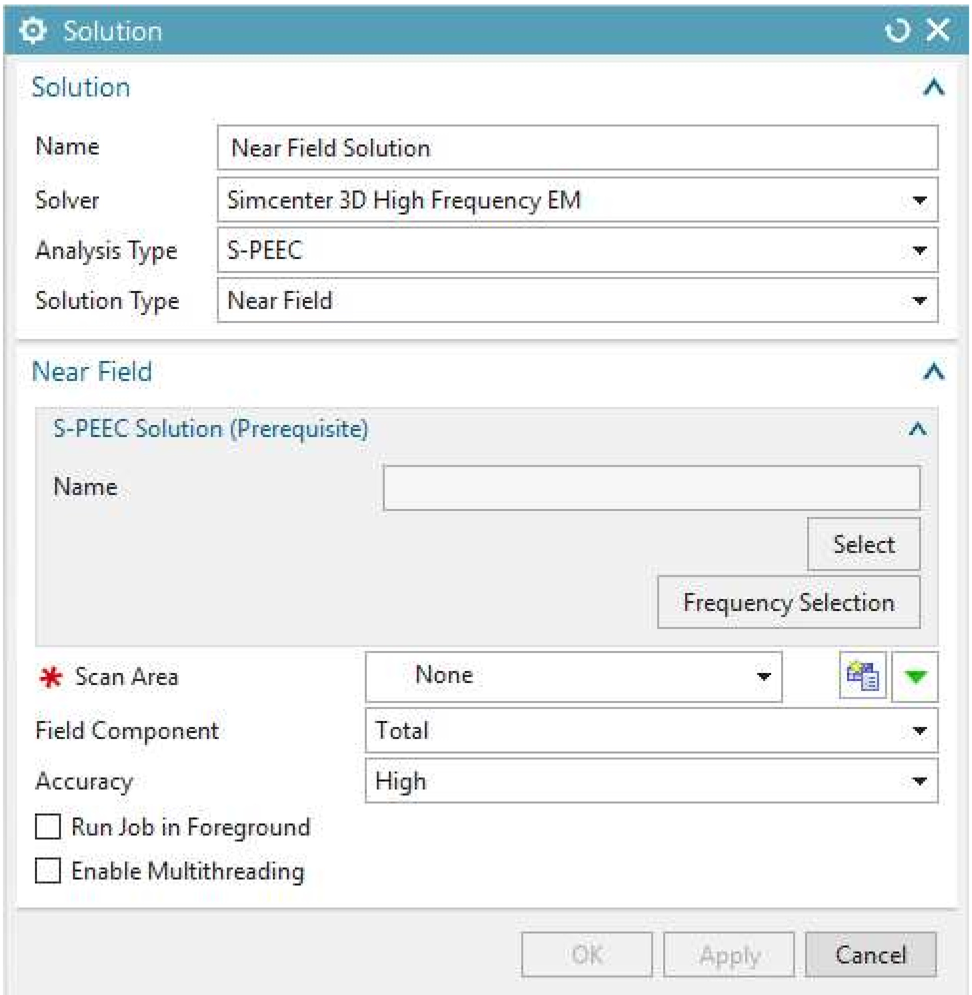


Solution

Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *S-PEEC*.
Solution Type	Specifies the solution type, in this case *Electric Potential*.
Induced Currents	
S-PEEC Solution (Prerequisite)	You can select and show a S-PEEC solution, as the prerequisite solution. • **Select**: you can select the prerequisite S-PEEC solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-40 – "S-PEEC - Electric Potential Solution" GUI fields

5.9.6.4 Near Field

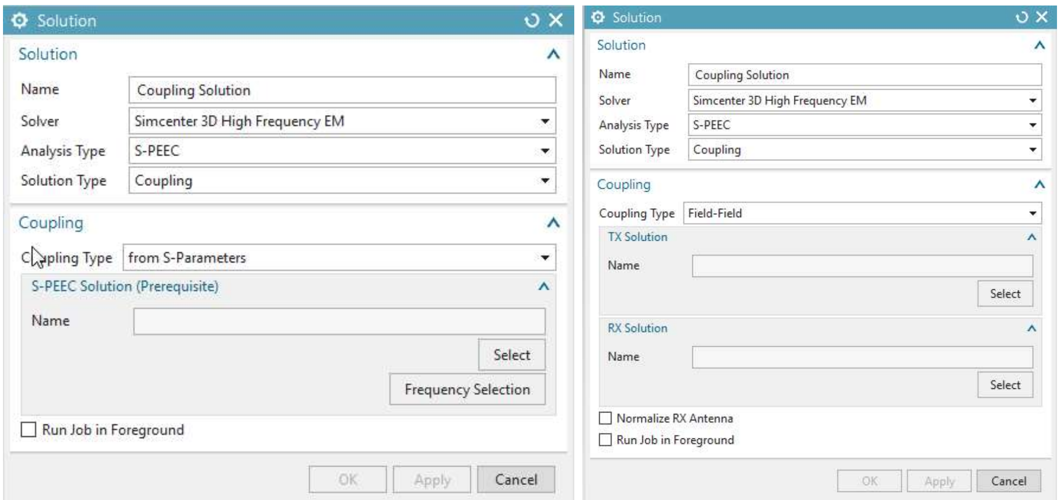


Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *S-PEEC*.
Solution Type	Specifies the solution type, in this case *Near Field*.
Near Field	
S-PEEC Solution (Prerequisite)	You can select and show a S-PEEC solution, as the prerequisite solution. • **Select**: you can select the prerequisite S-PEEC solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.

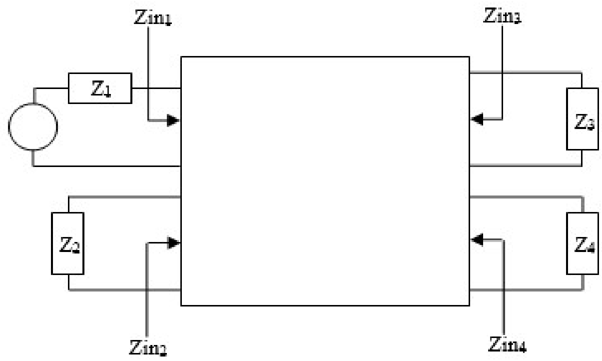
Scan Area	You can select one of the available near field scans (the analysis geometrical scan) or create a new one. For details see Par.5.3.2.
Field Component	You can select the field type to be computed. Available options are: • **Total**, the total field. • **Incident**, the field radiated by the source. • **Scattered**, the field scattered from the structure.
Accuracy	You can select the accuracy level. Available options are: **Medium**, **High**, **Very High**.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-41 – "S-PEEC - Near Field Solution" GUI fields

5.9.6.5 Coupling



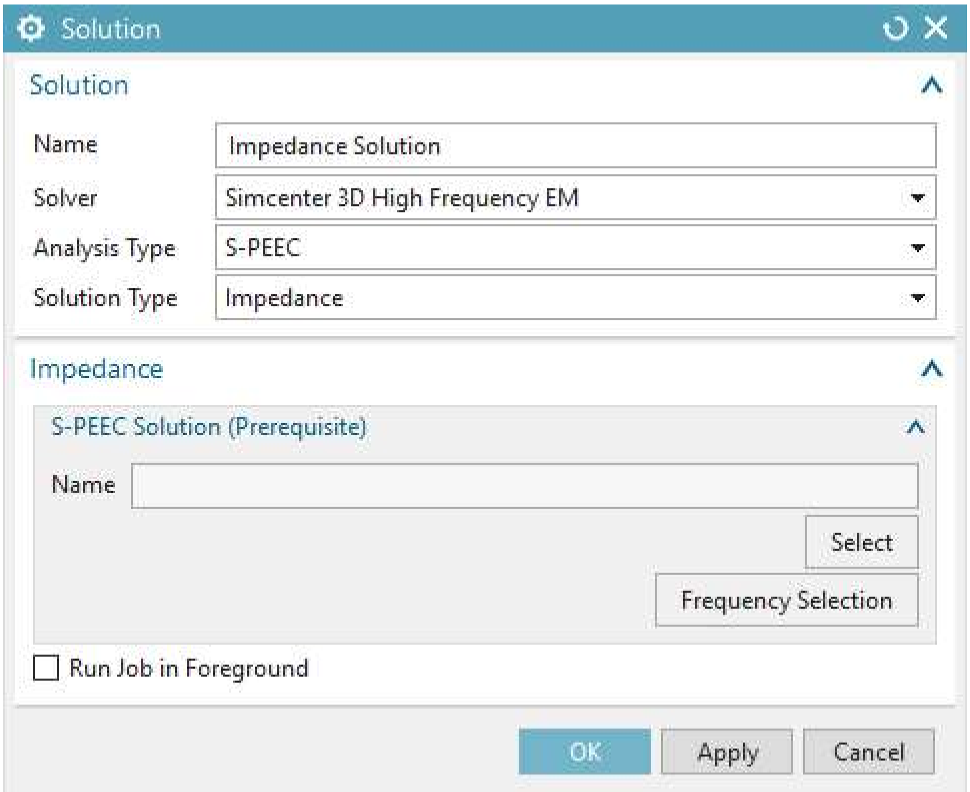
Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *S-PEEC*.
Solution Type	Specifies the solution type, in this case *Coupling*.
Coupling	
Coupling type	You can select coupling type.

	Available options are: • **from S-Parameters**: allows the calculation of the coupling between the model ports and evaluates four different types of inter-ports coupling:  ○ **Active Coupling,** i.e. coupling between the source port and the victim port, taking into account the real closing impedance conditions: ○ **T Coupling,** Transducer Coupling, i.e. coupling taking into account the actual loading conditions for the output ports; no power reflection is taken into account at the input port. ○ **A Coupling,** Available Input Power Coupling, i.e. coupling under condition of maximum power transfer (Zj=Z*inj) for the output ports; impedance mismatch at the input port is taken into account. ○ **Maximum Coupling**, i.e. inter-ports coupling under the condition of maximum power transfer (Zj=Z*inj). These definitions come from amplifier power gain definitions, see [BD45], par. 10.5. • **Field-Field**: evaluates coupling by performing a **"reaction integral" between two conformal near fields**. The two near fields **must be evaluated on the same geometrical scan** (i.e. on the same points).
S-PEEC Solution (Prerequisite)	Appears if the **Coupling type** is **from S-Parameters.** You can select and show a S-PEEC solution, as the prerequisite solution. • **Select**: you can select the prerequisite S-PEEC solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.

TX Solution	Appears if the **Coupling type** is **Field-Field.** You can select a TX solution, that is the “source” conformal near field.
RX Solution	Appears if the **Coupling type** is **Field-Field.** You can select a RX solution, that is the “victim” conformal near field.
Normalize RX Antenna	Appears if the **Coupling type** is **Field-Field.** The inter antenna coupling evaluated by means of the Field-Field integral is always normalized w.r.t. the power relevant to the Rx antenna near-field used in the calculation. The power value associated to a near-field is the input power (P_{in}) relevant to the antenna excited in the near-field evaluation. This power can differ from the radiated power (P_{rad}) for example when the antenna (or the platform) is loss affected. The coupling Field-Field integral will be normalized w.r.t.: • P_{rad} if the **Normalize Rx Antenna** is checked; • P_{in} otherwise.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.

Tab. 5-42 – “S-PEEC - Coupling Solution” GUI fields

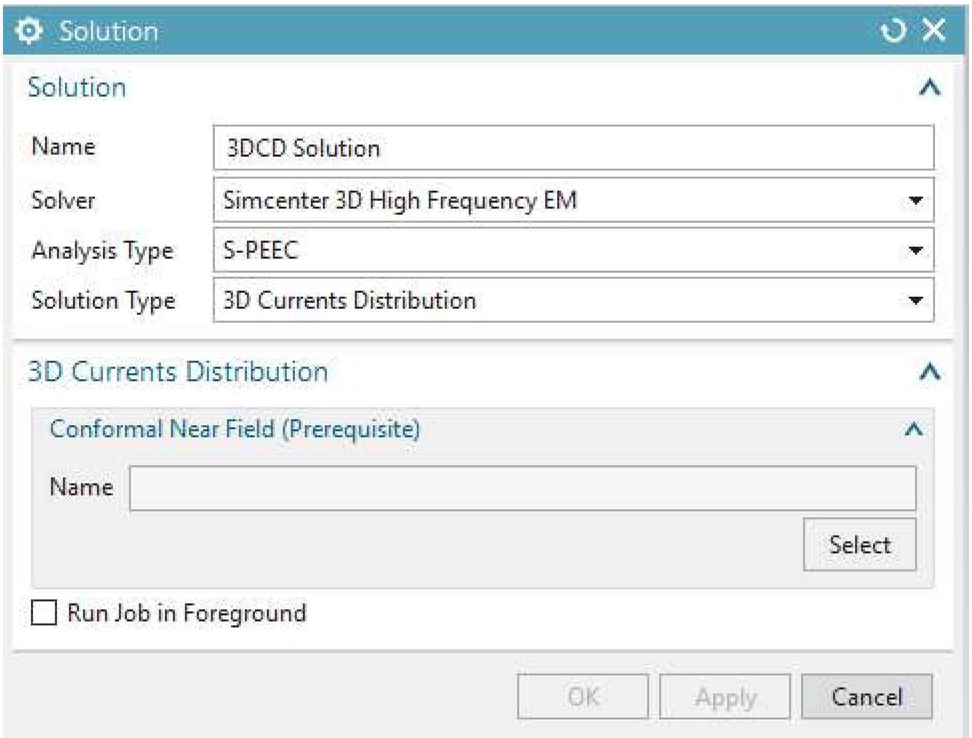
5.9.6.6 Impedance



Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *S-PEEC*.
Solution Type	Specifies the solution type, in this case *Impedance*.
Impedance	
S-PEEC Solution (Prerequisite)	You can select and show a S-PEEC solution, as the prerequisite solution. • **Select**: you can select the prerequisite S-PEEC solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.

Tab. 5-43 – “S-PEEC - Impedance Solution” GUI fields

5.9.6.7 3D Current Distribution



Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *S-PEEC*.
Solution Type	Specifies the solution type, in this case *3D Current distribution*.
3D Current distribution	
Conformal Near Field (Prerequisite)	You can select and show a Conformal Near Field solution, as the prerequisite solution.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.

Tab. 5-44 – "S-PEEC - 3D Current distribution Solution" GUI fields

5.10 UTD Analysis

5.10.1 Mesh type

Section 5.1 contains the description of the available mesh types for all analysis types.

5.10.2 Physical Property Tables

The following Physical Property Tables apply to *UTD* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Half Space* (5.2.1)
- *Multilayer* (5.2.2)
- *Material Layer* (5.2.3)
- *Tabulated Data* (5.2.4)
- *Material Tabulated* (Hidden 5.2.5)
- *Grid* (5.2.6)
- *Surface Impedance* (5.2.7)

5.10.3 Modelling Objects

The following Modeling Objects apply to *UTD* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Frequency Scan* (5.3.1)
- *Near Field Scan Area* (5.3.2)
- *Far Field Scan Area* (5.3.3)

5.10.4 Simulation Objects

The following Simulation Objects apply to *UTD* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Infinite Ground Plane* (5.5.2)

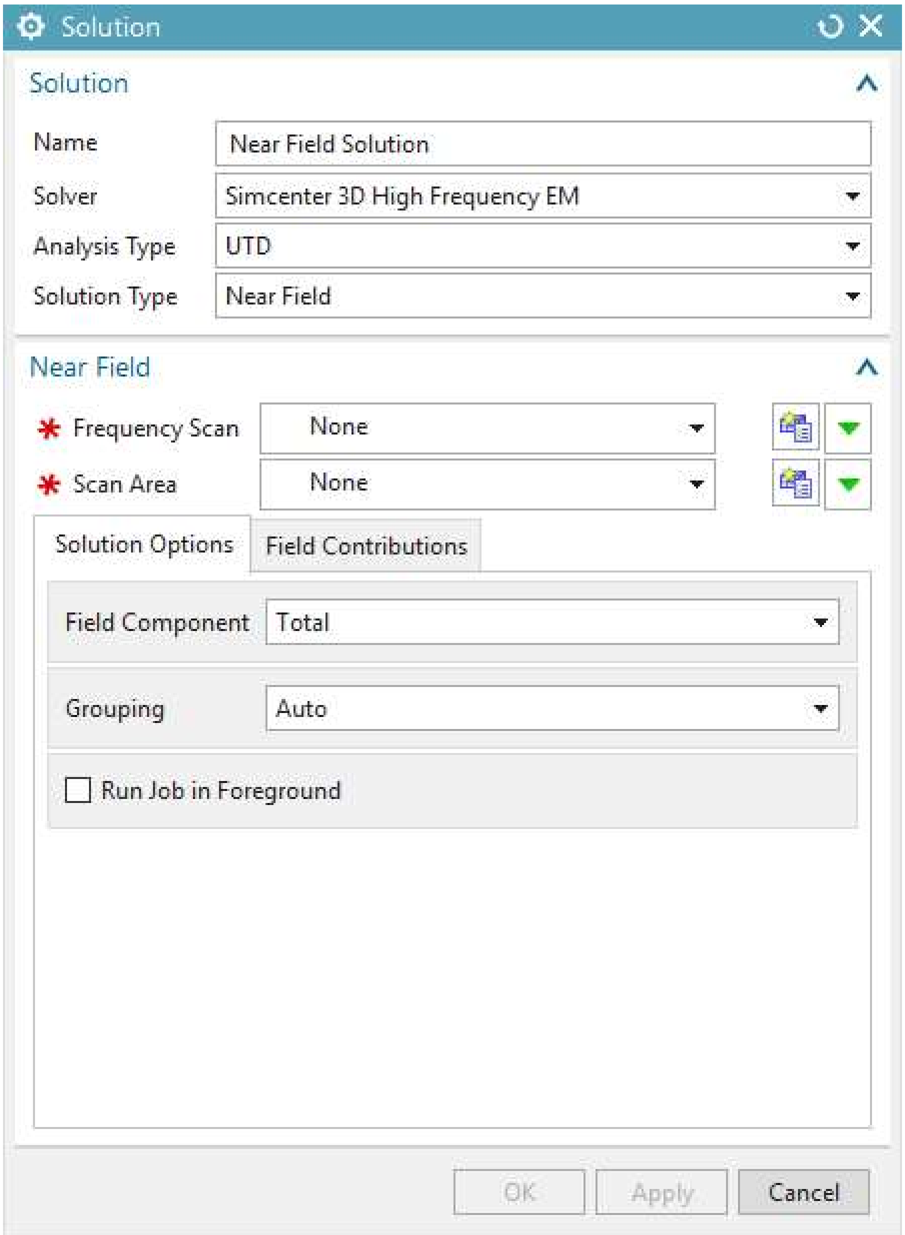
5.10.5 Loads

The following Loads apply to *UTD* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Synthetic Antenna Models* (5.6.2)
- *Magnetic Dipole* (5.6.3)
- *Electric Dipole* (5.6.4)

5.10.6 Solutions

5.10.6.1 Near Field



Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *UTD*.
Solution Type	Specifies the solution type, in this case *Near Field*.
Near Field	

Frequency scan	You can select one of the available frequency scans (the analysis frequencies) or create a new one. For details see Par. 5.3.1.
Scan Area	You can select one of the available near field scans (the analysis geometrical scan) or create a new one. For details see Par.5.3.2.
Solution Options	

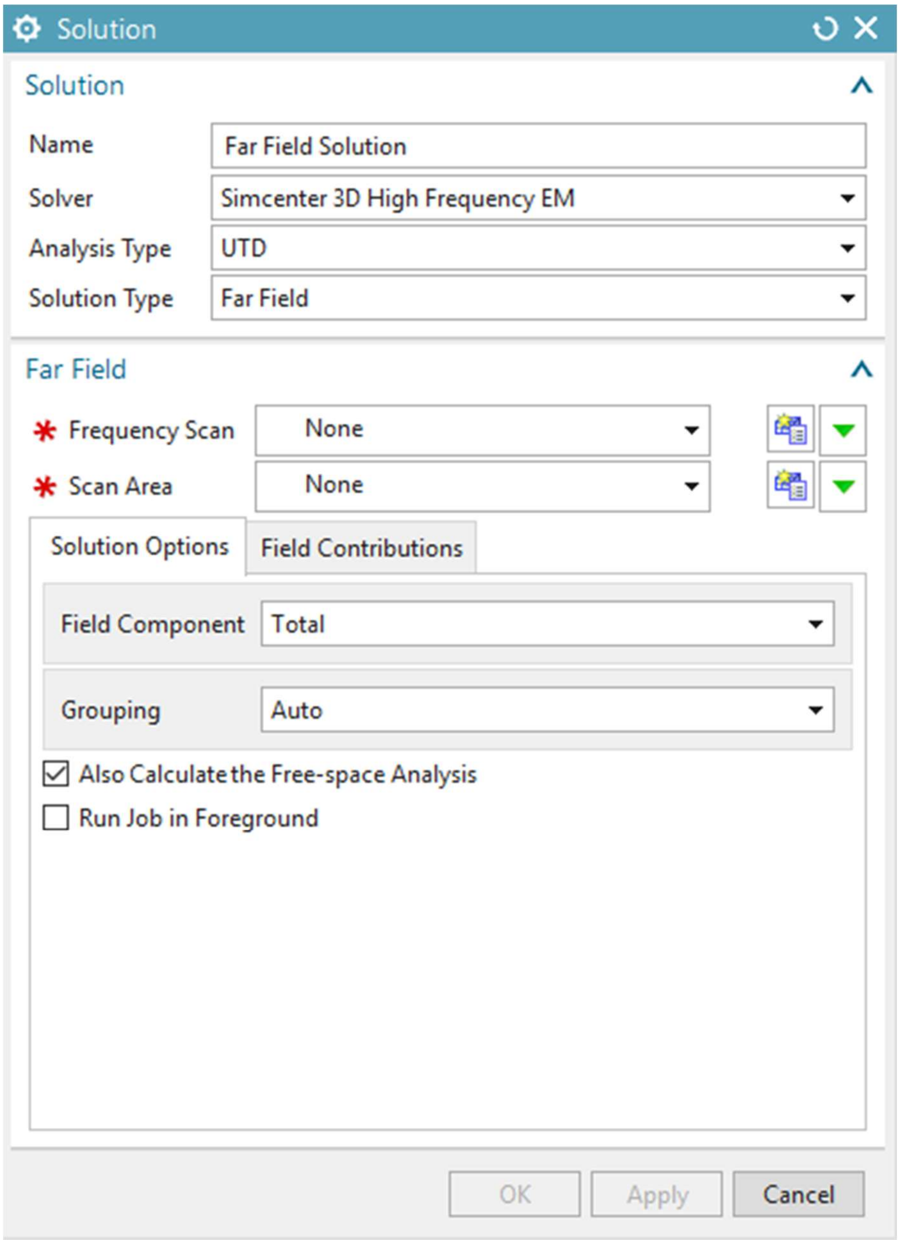
Field Component	You can select the field type to be computed. Available options are:
	• **Total**, the total field. The following output is evaluated with respect to **Direct** fields selection as follow:
	  o if the **Direct** contribution is selected:
	  $E_{out} = E_{dir} + E_{scatt}$
	  o if the **Direct** contribution is not selected:
	  $E_{out} = E_{scatt}$
	  Where E_{scatt} is the field due to all the accounted contributions (reflection/diffraction).
	• **Incident**, the field radiated by the source.
	• **Scattered**, the field scattered from the structure. The following output is evaluated:
	$$E_{out} = \begin{cases} E_{dir} + E_{scatt} - E_{source} & in\ \ view\ \ zone \\ E_{scatt} - E_{source} & in\ \ shadow\ \ zone \end{cases}$$
	  where E_{scatt} is the field due to all the accounted contributions (reflection/diffraction).
	Note: the direct contribution is always computed if **Scattered** is selected.

Grouping	You can group the equivalent currents constituting the 2D and 3D Current Distribution using several different criteria. Grouping sources means saving computational time. Available options are: • **None**: no grouping is applied. • **Auto**: automatic grouping depending on the source distribution in the space and its distance from the structure and the observer. • **All**: the currents constituting the source are grouped considering the same propagation path for all the single sources, applying a phase correction factor respect to the barycentre of the distribution considering a parallel-ray approximation.

Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Field Contribution	
Direct	Enables the computation of the direct contribution where: $$E_{dir} = \begin{cases} E_{source} & in \quad view \quad zone \\ 0 & in \quad shadow \quad zone \end{cases}$$ This contribution is accounted for in the output computation as detailed in the previous **Field Component** section.
Reflection from Plates	If checked, you can set the order of the multiple reflection contributions from plates. Note: multiple reflections are evaluated only for plates with coplanar normal.
Diffraction from Plates	If checked, you can set the order of the multiple diffraction contributions from plates.
Vertex Diffraction	Enables the Single diffraction from plate vertexes.
Hybrid Contribution from Plate	• **Plate-Edge/Wedge**: enables Double order contributions; it selects rays reflected from plates and then diffracted from wedges/edges. • **Plate-Vertex**: enables Double order contributions; it selects rays reflected from plates and then diffracted from plate vertexes.

Tab. 5-45 – "UTD - Near Field Solution" GUI fields

5.10.6.2 Far Field



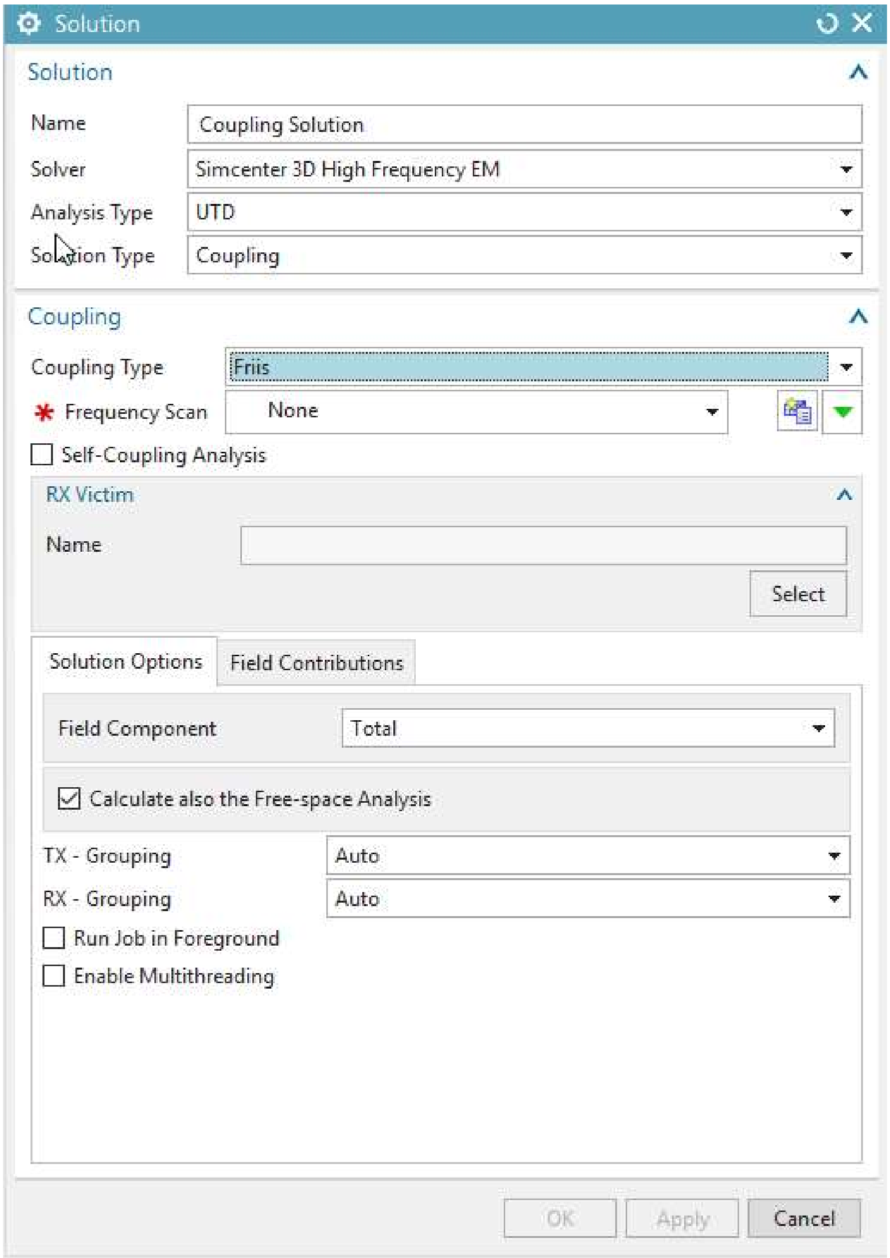
Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *UTD*.
Solution Type	Specifies the solution type, in this case *Far Field*.
Near Field	

Frequency scan	You can select one of the available frequency scans (the analysis frequencies) or create a new one. For details see Par.5.3.1.
Scan Area	You can select one of the available far field scans (the analysis geometrical scan) or create a new one. For details see Par.5.3.3.
Solution Options	
Field Component	You can select the field type to be computed. Available options are: • **Total**, the total field. The following output is evaluated with respect to **Direct** field selection as follows: o if the **Direct** contribution is selected: $E_{out} = E_{dir} + E_{scatt}$ o if the **Direct** contribution is not selected: $E_{out} = E_{scatt}$ Where E_{scatt} is the field due to all the accounted contributions (reflection/diffraction). • **Incident**, the field radiated by the source. • **Scattered**, the field scattered from the structure. The following output is evaluated: $E_{out} = \begin{cases} E_{dir} + E_{scatt} - E_{source} & in \quad view \quad zone \\ E_{scatt} - E_{source} & in \quad shadow \quad zone \end{cases}$ where E_{scatt} is the field due to all the accounted contributions (reflection/diffraction). Note: the direct contribution is always computed if **Scattered** is selected.
Grouping	You can group the equivalent currents constituting the 2D and 3D Current Distribution using several different criteria. Grouping sources means saving computational time. Available options are: • **None**: no grouping is applied. • **Auto**: automatic grouping depending on the source distribution in the space and its distance from the structure and the observer. • **All**: the currents constituting the source are grouped considering the same propagation path for all the single sources, applying a phase correction factor with respect to the barycentre of the distribution considering a parallel-ray approximation.

Also calculate the Free-space Analysis	If selected, the solution is run also using only the input Loads, without using the mesh model. In the post-processing phase, a comparison function between the regular run and the free-space one will be available.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Field Contribution	
Direct	Enables the computation of the direct contribution where: $E_{dir} = \begin{cases} E_{source} & in \quad view \quad zone \\ 0 & in \quad shadow \quad zone \end{cases}$ This contribution is accounted for in the output computation as detailed in the previous **Field Component** section.
Reflection from Plates	If checked, you can set the order of the multiple reflection contributions from plates. Note: Multiple reflections are evaluated only for plates with coplanar normal.
Diffraction from Plates	If checked, you can set the order of the multiple diffraction contributions from plates.
Vertex Diffraction	Enables the Single diffraction from plate vertexes.
Hybrid Contribution from Plate	• **Plate-Edge/Wedge**: enables Double order contributions. It selects rays reflected from plates and then diffracted from wedges/edges. • **Plate-Vertex**: enables Double order contributions. It selects rays reflected from plates and then diffracted from plate vertexes.

Tab. 5-46 – “UTD - Far Field Solution” GUI fields

5.10.6.3 Coupling



Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *UTD*.
Solution Type	Specifies the solution type, in this case *Coupling*.
Coupling	
Coupling type	You can select coupling type. Available choices are:

	• **Field-Currents**: evaluates coupling between one transmitting (TX) antenna and one receiving (RX) or victim antenna. Any antenna model can be used for the TX antenna, while only 2D Current Distribution can be chosen for the RX antenna. The code will compute the reaction between the TX antenna radiated field and the current distribution of the RX antenna. • **Friis**: evaluates coupling between one transmitting (TX) antenna and one receiving (RX) or victim antenna. Any antenna model can be used for both the TX and the RX antenna. The code will evaluate the coupling using the Friis transmission formula, so the RX source (the single constituting source for current distribution models) should be in the far-field region with respect to the structure and the TX source. • **Field-Field**: evaluates coupling by performing a **"reaction integral" between two conformal near fields**. The two near fields **must be evaluated on the same geometrical scan** (i.e. on the same points).
Frequency scan	You can select one of the available frequency scans (the analysis frequencies) or create a new one. For details see Par. 5.3.1.
Self-Coupling Analysis	Appears if the **Coupling type** is **Field-Currents** or **Friis.** If set, source and victim are the same antenna.
RX Victim	Only if the **Coupling type** is **Field-Currents** or **Friis** and **Self-Coupling Analysis** is not selected. You can select a RX victim source from the available source types in the Load Container (see Par.5.6 for details): • Magnetic dipoles; • Electric dipoles; • Synthetic Antenna models.
TX Solution	Appears if the **Coupling type** is **Field-Field.** You can select a TX solution, that is the "source" conformal near field.
RX Solution	Appears if the **Coupling type** is **Field-Field.** You can select a RX solution, that is the "victim" conformal near field.
Normalize RX Antenna	Appears if the **Coupling type** is **Field-Field.**

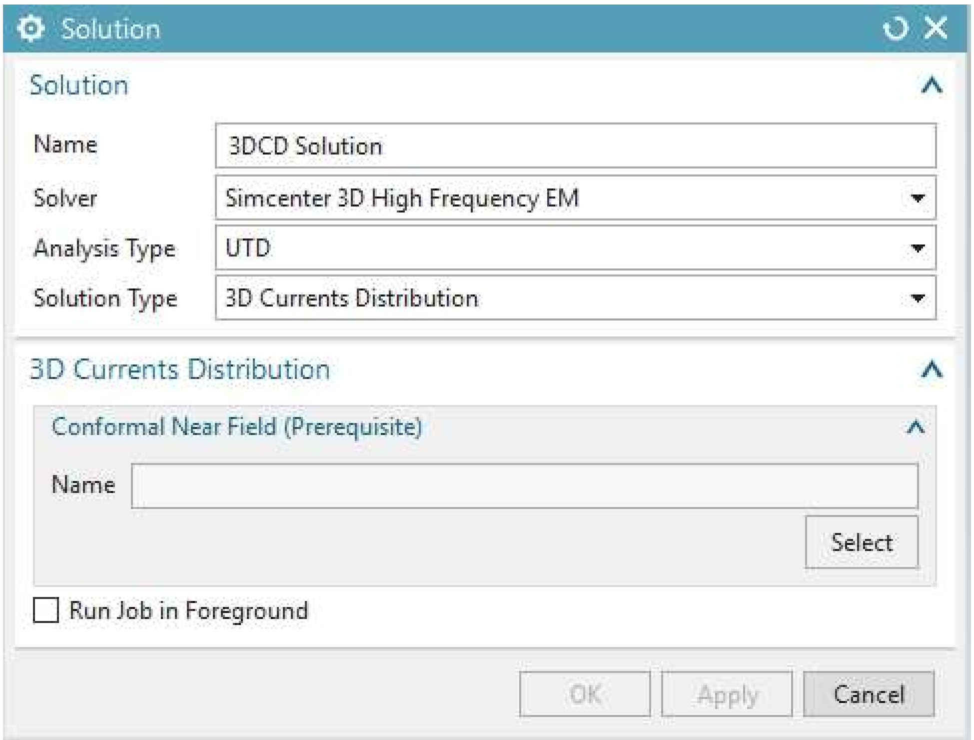
	The inter antenna coupling evaluated using the Field-Field integral is always normalized w.r.t. the power relevant to the Rx antenna near-field used in the calculation. The power value associated to a near-field is the input power (P_{in}) relevant to the antenna excited in the near-field evaluation. This power can differ from the radiated power (P_{rad}) for example when the antenna (or the platform) is loss affected. The coupling Field-Field integral will be normalized w.r.t.: • P_{rad} if the **Normalize Rx Antenna** is checked; • P_{in} otherwise.
Solution Options	
Field Component	Only if the **Self-Coupling Analysis** is not selected and if the **Coupling Type** is not **Field-Field**. You can select the field type to be computed. Available options are: • **Total**, the total field. The following output is evaluated with respect to **Direct** fields selection as follow: o if the **Direct** contribution is selected: $E_{out} = E_{dir} + E_{scatt}$ o if the **Direct** contribution is not selected: $E_{out} = E_{scatt}$ Where E_{scatt} is the field due to all the accounted for contributions (reflection/diffraction). • **Incident**, the field radiated by the source. • **Scattered**, the field scattered from the structure. The following output is evaluated: $E_{out} = \begin{cases} E_{dir} + E_{scatt} - E_{source} & \text{in view zone} \\ E_{scatt} - E_{source} & \text{in shadow zone} \end{cases}$ where E_{scatt} is the field due to all the accounted for contributions (reflection/diffraction). <u>Note</u>: the direct contribution is always computed if **Scattered** is selected.
Also calculate the Free-space Analysis	Only if the **Self-Coupling Analysis** is not selected and if the **Coupling Type** is not **Field-Field**. If selected, the solution is run also using only the input Loads, without using the mesh model. In the post-processing phase, a comparison function between the regular run and the free-space one will be available.

TX - Grouping	Appears only if the **Coupling Type** is not **Field-Field.** You can group the equivalent currents constituting the 2D and 3D Current Distribution using several different criteria. Grouping sources means saving computational time. Available options are: • **None**: no grouping is applied. • **Auto**: automatic grouping depending on the source distribution in the space and its distance from the structure and the observer. • **All**: the currents constituting the source are grouped considering the same propagation path for all the single sources, applying a phase correction factor with respect to the barycentre of the distribution considering a parallel-ray approximation.
RX - Grouping	Only if **Coupling Type** is **Friis** and **Self-Coupling Analysis** is not selected. You can group the equivalent currents constituting the 2D and 3D Current Distribution using several different criteria. Grouping sources means saving computational time. Available options are: • **None**: no grouping is applied. • **Auto**: automatic grouping depending on the source distribution in the space and its distance from the structure and the observer. • **All**: the currents constituting the source are grouped considering the same propagation path for all the single sources, applying a phase correction factor with respect to the barycentre of the distribution considering a parallel-ray approximation.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	Appears only if the **Coupling Type** is not **Field-Field.** If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.
Field Contribution (Only if the Self-Coupling Analysis or the Field-Field analysis types are selected)	
Direct	Enables the computation of the direct contribution where: $E_{dir} = \begin{cases} E_{source} & in \quad view \quad zone \\ 0 & in \quad shadow \quad zone \end{cases}$

	This contribution is accounted for in the output computation as detailed in the previous **Field Component** section.
Reflection from Plates	If checked, you can set the order of the multiple reflection contributions from plates. Note: Multiple reflections are evaluated only for plates with coplanar normal.
Diffraction from Plates	If checked, you can set the order of the multiple diffraction contributions from plates.
Vertex Diffraction	Enables the Single diffraction from plate vertexes.
Hybrid Contribution from Plate	• **Plate-Edge/Wedge**: enables Double order contributions. It selects rays reflected from plates and then diffracted from wedges/edges. • **Plate-Vertex**: enables Double order contribution. It selects rays reflected from plates and then diffracted from plate vertexes.

Tab. 5-47 – "UTD –Coupling Solution" GUI fields

5.10.6.4 3D Current Distribution



Solution	
Name	Specifies the name of the solution.

Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *UTD*.
Solution Type	Specifies the solution type, in this case *3D Current distribution*.
3D Current distribution	
Conformal Near Field (Prerequisite)	You can select and show a Conformal Near Field solution, as the prerequisite solution.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.

Tab. 5-48 – "UTD - 3D Current distribution Solution" GUI fields

5.11 Iterative PO Analysis

5.11.1 Mesh type

Section 5.1 contains the description of the available mesh types for all analysis types.

5.11.2 Physical Property Tables

The following Physical Property Tables apply to *Iterative PO* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Half Space* (5.2.1)
- *Multilayer* (5.2.2)
- *Material Layer* (5.2.3)
- *Tabulated Data* (5.2.4)
- *Material Tabulated* (Hidden 5.2.5)
- *Grid* (5.2.6)
- *Surface Impedance* (5.2.7)

5.11.3 Modelling Objects

The following Modeling Objects apply to *Iterative PO* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Frequency Scan* (5.3.1)
- *Near Field Scan Area* (5.3.2)
- *Far Field Scan Area* (5.3.3)

5.11.4 Simulation Objects

The following Simulation Objects apply to *Iterative PO* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Infinite Ground Plane* (5.5.2)

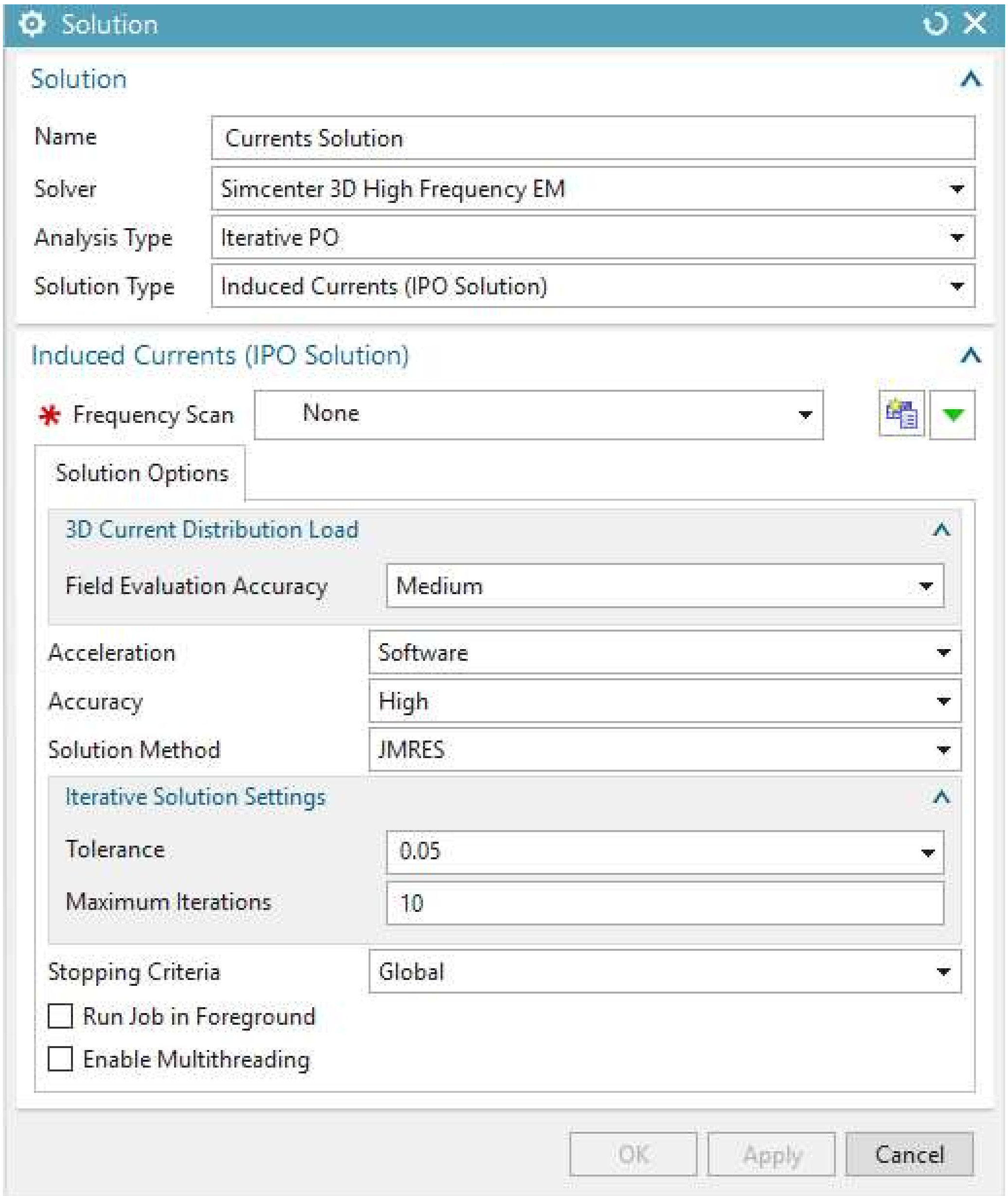
5.11.5 Loads

The following Loads apply to *Iterative PO* Analyses in the *Simcenter 3D High Frequency EM* environment:

- *Synthetic Antenna Models* (5.6.2)
- *Magnetic Dipole* (5.6.3)
- *Electric Dipole* (5.6.4)
- *EM Plane Wave* (5.6.5)

5.11.6 Solutions

5.11.6.1 Induced Currents (IPO Solution)



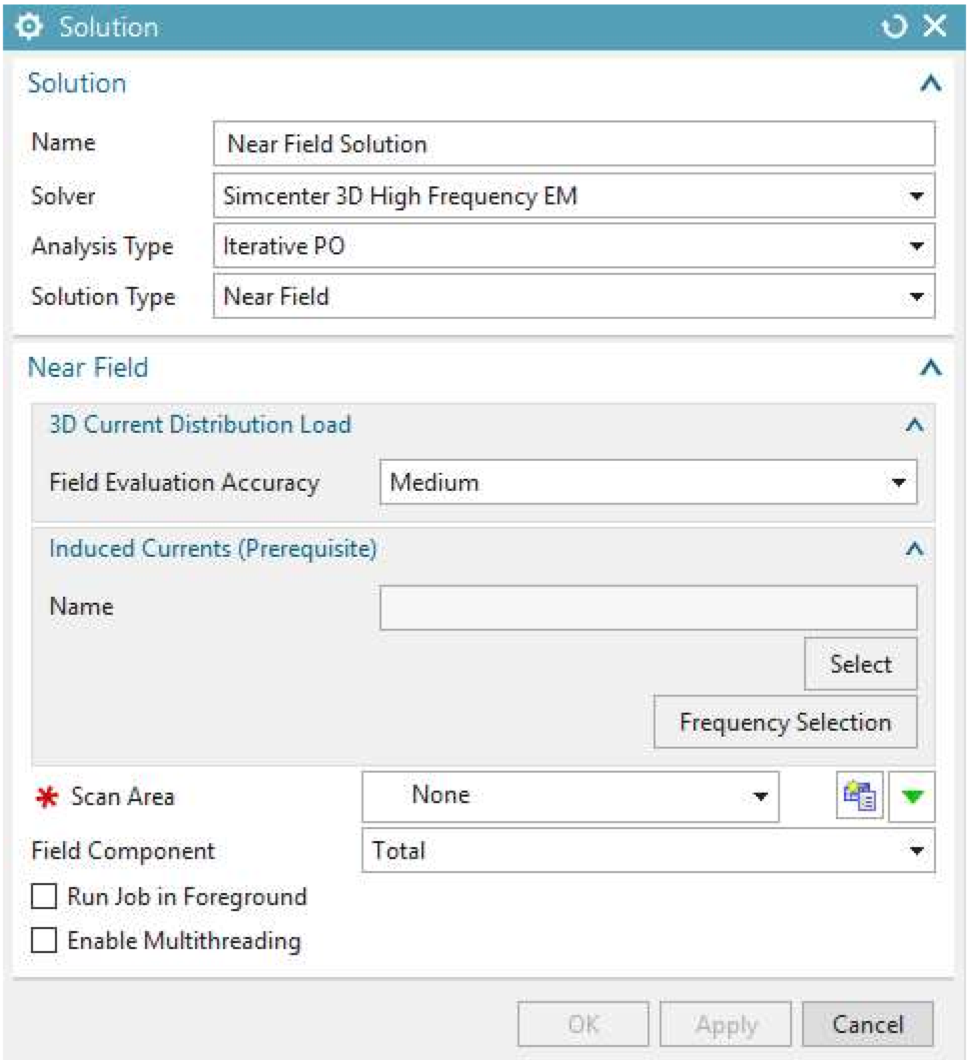
Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *Iterative PO*.
Solution Type	Specifies the solution type, in this case *Induced Currents (IPO Solution)*.
Induced Currents (IPO Solution)	

Frequency scan	You can select one of the available frequency scans (the analysis frequencies) or create a new one. For details see Par.5.3.1.
Solution Options	
3D Current Distribution Load	You can define the Field Evaluation Accuracy from the source if a 3D Current Distribution is selected as the Load. Available options are: • Medium; • High; • Very High.
Acceleration	Specifies the acceleration algorithm. Available algorithms are: • **None**; • **Software**; • **Hardware**. Selecting the "Software" option, the Fast Far-Field Approximation (FaFFA) algorithm is used to accelerate the calculation of the induced currents at each step of the iterative procedure as well as the PO currents produced by the source. The FaFFA algorithm is based on a domain decomposition of the structure. The interactions between elements belonging to blocks at a distance of more than a certain threshold are not computed directly but through a grouping of the field contributions. The threshold is automatically defined by the algorithm. It can be demonstrated that the overall computational complexity of the algorithm is O ($N^{3/2}$) where N is the number of mesh elements. Selecting the "Hardware" option, the NVIDIA GPU (CUDA capable devices) calculation platform is used.
GPU Selection	Only available if the acceleration algorithm is **Hardware**. You can select the GPUs (Graphics Processing Unit) that will be used in the simulation from a list of available units.
Accuracy	Only available if the acceleration algorithm is **Software** or **Hardware**. You can define the evaluation accuracy. Available options are: • Medium (Software acceleration only); • High; • Very High.
Solution Method	Available solution methods are [BD35]:

	• **JMRES**: (Jacobi Minimal RESidual) provides in most of the cases better convergence behaviour than Jacobi: it uses two degrees of freedom to minimize the residual error at each step. • **Jacobi**: it use the Jacobi algorithm.
Relaxation Factor	This field is enabled only if **Jacobi** Solution Method is selected. Recommended value is 0.7. The lower the value is the more likely to converge method is, but with a greater number of iterations. Allowed values are within the following range (0;1].
Iterative Solution Settings	You can assign control parameters of the iterative methods implemented in the IPO solver. • **Tolerance**: sets the tolerance for the iterative method. The iterative method stops when the residual is lower than this value. • **Maximum Iterations**: sets the maximum number of iterations of the iterative method. The iterative method stops when the number of iterations exceeds this value.
Stopping Criteria	Defines the criteria for the evaluation of the residual. Available options are: • **Global:** the residual is defined as a global parameter considering all the elements of the mesh. The iterative method stops when such residual is lower than the tolerance. This is the less demanding criteria. • **Local:** the residual is defined as a local parameter (i.e. at single mesh element level). The iterative method stops when all the residuals are lower than the tolerance. This is the most stringent criteria. • **Max Order:** The iterative method stops when the maximum number of iterations is reached regardless of the residual value (Global criteria).
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-49 – "Iterative PO – Induced Currents Solution" GUI fields

5.11.6.2 Near Field

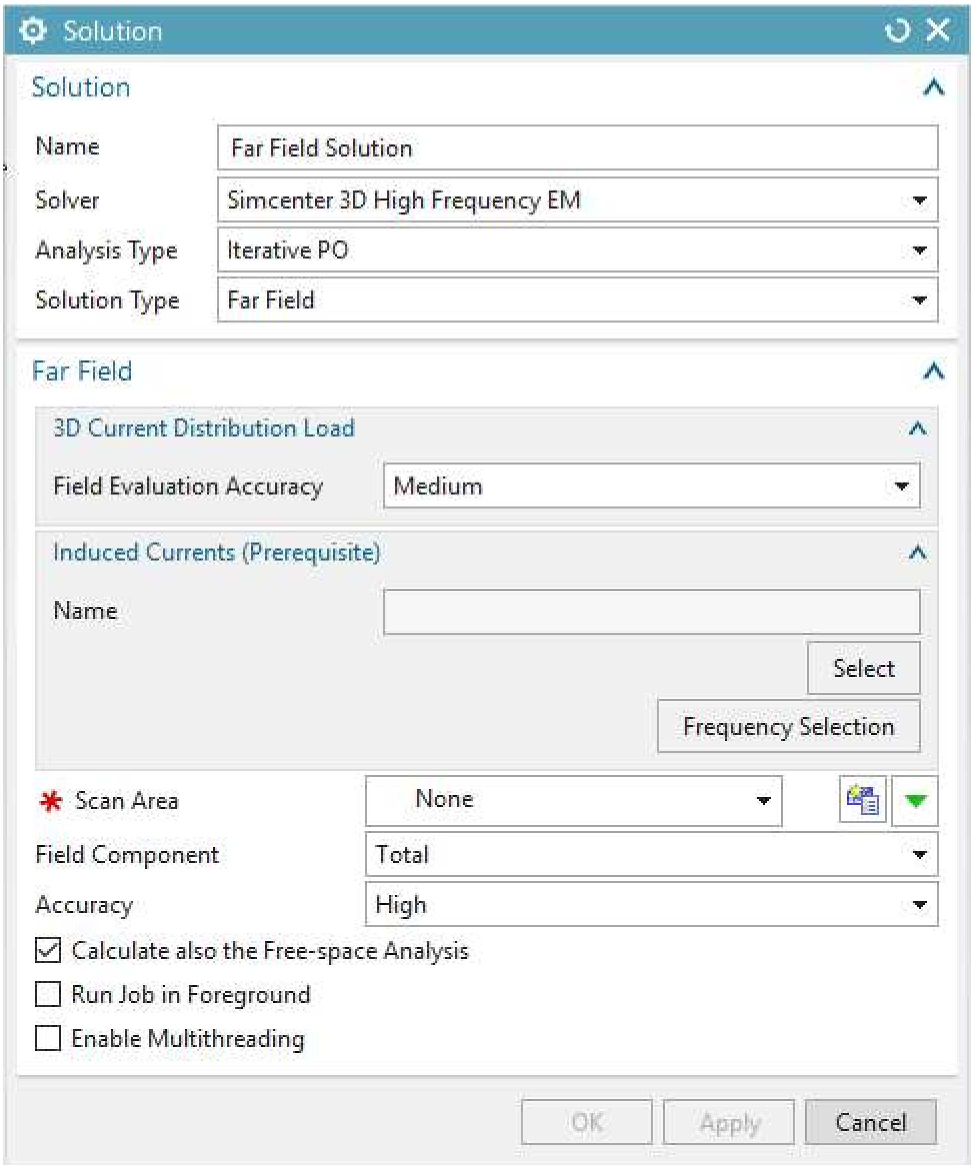


Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *Iterative PO*.
Solution Type	Specifies the solution type, in this case *Near Field*.
Near Field	
3D Current Distribution Load	You can define the Field Evaluation Accuracy from the source if 3D Current Distribution is selected as the Load. Available options are: • Medium; • High; • Very High.

Induced Currents (Prerequisite)	You can select and show a Induced Currents solution, as the prerequisite solution. • **Select**: you can select the prerequisite Induced Currents solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Scan Area	You can select one of the available near field scans (the analysis geometrical scan) or create a new one. For details see Par.5.3.2.
Field Component	You can select the field type to be computed. Available options are: • **Total**, the total field. • **Incident**, the field radiated by the source. • **Scattered**, the field scattered from the structure.
Accuracy	Appears if the Scan Area is a conformal scan. You can select the accuracy level. Available options are: **Medium**, **High**, **Very High**.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-50 – "Iterative PO - Near Field Solution" GUI fields

5.11.6.3 Far Field

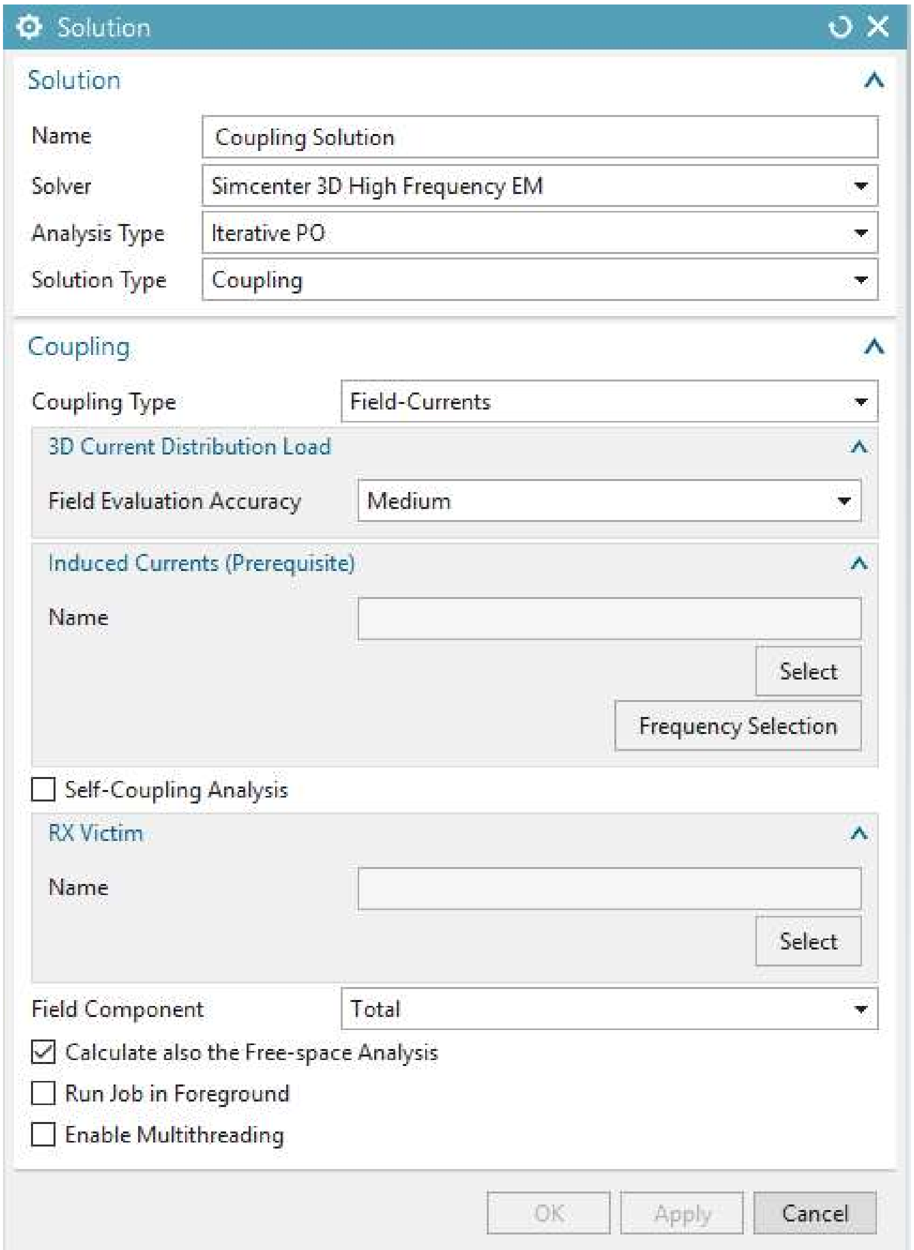


Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *Iterative PO*.
Solution Type	Specifies the solution type, in this case *Far Field*.
Far Field	
3D Current Distribution Load	You can define the Field Evaluation Accuracy from the source if a 3D Current Distribution is selected as the Load. Available options are: • Medium;

	• High; • Very High.
Induced Currents (Prerequisite)	You can select and show a Induced Currents solution, as the prerequisite solution. • **Select**: you can select the prerequisite Induced Currents solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Scan Area	You can select one of the available far field scans (the analysis geometrical scan) or create a new one. For details see Par.5.3.3.
Field Component	You can select the field type to be computed. Available options are: • **Total**, the total field. • **Incident**, the field radiated by the source. • **Scattered**, the field scattered from the structure.
Accuracy	You can select the accuracy level. Available options are: **Medium**, **High**, **Very High**.
Calculate also the Free-space Analysis	If selected, the solution is run also using only the input Loads, without using the mesh model. In the post-processing phase a comparison function between the regular run and the free-space one will be available.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-51 – "Iterative PO - Far Field Solution" GUI fields

5.11.6.4 Coupling



Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *Iterative PO*.
Solution Type	Specifies the solution type, in this case *Coupling*.
Coupling	
Coupling type	You can select coupling type. Available choices are:

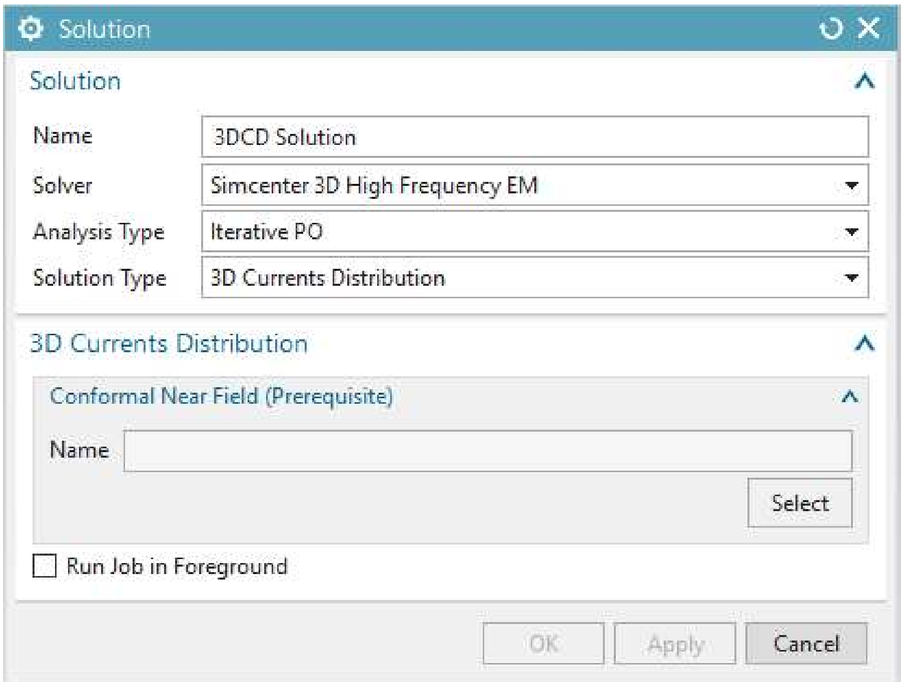
	• **Field-Currents**: evaluates coupling between one transmitting (TX) antenna and one receiving (RX) or victim antenna. Any antenna model can be used for the TX antenna, while only 2D Current Distribution can be chosen for the RX antenna. The code will compute the reaction between the TX antenna radiated field and the current distribution of the RX antenna. • **Friis**: evaluates coupling between one transmitting (TX) antenna and one receiving (RX) or victim antenna. Any antenna model can be used for both the TX and the RX antenna. The code will evaluate the coupling using the Friis transmission formula, so the RX source (the single constituting source for current distribution models) should be in the far-field region with respect to the structure and the TX source. • **Field-Field**: evaluates coupling by performing a **“reaction integral” between two conformal near fields**. The two near fields **must be evaluated on the same geometrical scan** (i.e. on the same points).
3D Current Distribution Load	You can define the Field Evaluation Accuracy from the source if a 3D Current Distribution is selected as the Load for the TX antenna. Available options are: • Medium; • High; • Very High.
Induced Currents (Prerequisite)	You can select and show an Induced Currents solution, as the prerequisite solution. • **Select**: you can select the prerequisite Induced Currents solution. • **Frequency selection**: you can select the analysis frequencies as a subset of frequencies of the selected prerequisite solution.
Self-Coupling Analysis	Appears if the **Coupling type** is **Field-Currents** or **Friis.** If set, source and victim are the same antenna.
RX Victim	Only if the **Coupling type** is **Field-Currents** or **Friis** and **Self-Coupling Analysis** is not selected. You can select a RX victim source, from the available source types in the Load Container (see Par.5.6 for details):

	• Magnetic dipoles; • Electric dipoles; • Synthetic models.
TX Solution	Appears if the **Coupling type** is **Field-Field.** You can select a TX solution, that is the “source” conformal near field.
RX Solution	Appears if the **Coupling type** is **Field-Field.** You can select a RX solution, that is the “victim” conformal near field.
Normalize RX Antenna	Appears if the **Coupling type** is **Field-Field.** The inter antenna coupling evaluated using the Field-Field integral is always normalized w.r.t. the power relevant to the Rx antenna near-field used in the calculation. The power value associated to a near-field is the input power (P_{in}) relevant to the antenna excited in the near-field evaluation. This power can differ from the radiated power (P_{rad}), for example, when the antenna (or the platform) is loss affected. The coupling Field-Field integral will be normalized w.r.t.: • P_{rad} if the **Normalize Rx Antenna** is checked; • P_{in} otherwise.
Solution Options	
Field Component	Only if **Self-Coupling Analysis** is not selected and if the **Coupling Type** is not **Field-Field**. You can select the field type to be computed. Available options are: • **Total**, the total field (incident + scattered). • **Incident**, the field radiated by the source. • **Scattered**, the field radiated by the currents induced on the structure.
Calculate also the Free-space Analysis	Only if the **Self-Coupling Analysis** is not selected and if the **Coupling Type** is not **Field-Field**. If selected, the solution is also run using only the input Loads, without using the mesh model. In the post-processing phase, a comparison function between the regular run and the free-space one will be available.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.
Enable Multithreading	Appears only if the **Coupling Type** is not **Field-Field.**

	If selected, the solution run is parallelized on the specified number of threads.
Number of Execution Threads	If the **Enable Multithreading** option is selected, specifies the number of execution threads.

Tab. 5-52 – "Iterative PO – Coupling Solution" GUI fields

5.11.6.5 3D Current Distribution



Solution	
Name	Specifies the name of the solution.
Solver	Specifies the solver for the solution.
Analysis Type	Specifies the analysis, in this case *Iterative PO*.
Solution Type	Specifies the solution type, in this case *3D Current distribution*.
3D Current distribution	
Conformal Near Field (Prerequisite)	You can select and show a Conformal Near Field solution, as the prerequisite solution.
Run Job in Foreground	If selected, the solution is run in the foreground. If cleared, the job is run in the background.

Tab. 5-53 – "Iterative PO - 3D Current distribution Solution" GUI fields