HarPE™ Technical Brief

INTRODUCTION

HarPE™ is dedicated to FET parameter extraction, device simulation and model building. Large- and small-signal data can be processed.

HarPE is the world's first software system exploiting harmonic data for accurate nonlinear FET model parameter extraction, while offering a comprehensive approach to FET device simulation.

Using HarPE, microwave circuit designers can make the most effective use of Libra, Touchstone, Microwave Harmonica and Super-Compact.

MEASUREMENT DATA

HarPE accepts device simulations or measurements in the form of:

- power spectra
- waveforms
- small-signal S-parameters
- DC bias data

HarPE features a flexible and intelligent input data format which is readily tailored by the user to accommodate raw data in different forms.

BUILT-IN DEVICE MODELS

HarPE supports three popular FET models:

- the Curtice and Ettenberg model
- the Materka and Kacprzak model
- the Raytheon (Statz *et al.*) model

Device models are implemented consistently with Libra and Microwave Harmonica, into whose input files you can directly paste HarPE's output of FET parameters.

CREATE OR CUSTOMIZE MODELS

HarPE allows you to define your own device models using Fortran-like expressions.

Alternatively, you may customize or alter the built-in library models to reflect any functional dependence of any defined model parameters. Your customized models enjoy all computational options of HarPE.

Combinations of algebraic operations and standard functions are permitted, including:

```
EXP, LOG, LOG10, SQRT, ABS,
SIN, COS, TAN, ASIN, ACOS, ATAN,
SINH, COSH, TANH
```

A simple mechanism for evaluating conditional expressions is also provided.

Using these mathematical expressions, you can describe frequency dependence, temperature dependence, bias dependence, etc. For example, to describe a simple bias dependence of drain-source conductance GDS, you may use

```
GDS = GDS0 + B * VG + C * VD
```

where VG and VD are gate and drain bias voltages and B and C are user-specified coefficients.

HarPE enables you to create models more simply and more effectively than you ever imagined possible! HarPE is the most versatile tool ever created for designing and gaining insight into proposed FET models.

Within about a day you should be able to create and test any new model. Consequently, HarPE is one of the most cost effective tools your company could invest in.

PARAMETER EXTRACTION FROM S-PARAMETERS

HarPE extracts large-signal model parameters from small-signal S-parameter and/or DC measurement data. HarPE simultaneously fits multi-bias, multi-frequency responses. The nonlinear characteristics automatically and smoothly links the model at different bias points.

EXTRACTION FROM HARMONIC DATA

As a unique feature, HarPE offers a truly nonlinear FET parameter extraction procedure which utilizes spectrum measurements, including DC bias information and power output at different harmonics measured under RF large-signal excitations.

The HarPE criterion for parameter extraction is that the computed and measured responses must simultaneously and optimally match at DC, fundamental frequency and higher harmonics. The HarPE approach ensures the most reliable solutions of large-signal model parameters for subsequent circuit simulation with circuit simulators.

HarMONIZE LARGE- AND SMALL-SIGNAL SIMULATIONS!

HarPE works effectively as a device simulator with or without measurement data. HarPE provides graphical displays and numerical outputs and comparisons for

- DC IV curves
- harmonic responses vs. sweep of fundamental input powers
- harmonic responses vs. sweep of fundamental excitation frequencies
- waveform responses of voltages and currents
- spectrum responses of powers, voltages and currents
- Smith chart and polar plots of S-parameters

HarPE seamlessly links large- and small-signal responses in a unified manner. The same circuit file and the same corresponding large-signal model description are used to compute S-parameter responses, DC IV curves, spectrum and waveform responses. For example, you can use a device model optimally extracted from S-parameter data to simulate spectrum and waveform responses.

HarPE IS TRULY USER-FRIENDLY!

HarPE is so simple to use, you are immediately productive. You can quickly create models and explore them faster than you ever thought possible. HarPE may actually pay for itself in the first month of use!

The following features distinguish the most user-friendly CAD system ever offered to solve microwave simulation problems!

- Versatile, user-friendly input file.
- Built-in dual-window, dual-file full screen editor with search-replace, cut-paste, on-line macro, undo features and more.
- Fully integrated file manager, file parser and screen editor. The location of any error detected during file parsing is automatically highlighted by the editor.
- Logical, easy to understand, easy to use, menu-driven and window-driven commands with on-line help for every command.
- On-line User's Manual. You can add your own comments to the Manual, or copy syntax definitions and examples in the Manual and paste them directly into the circuit file, or define and search for arbitrary keywords (dynamic indexing).
- Macro file capability facilitates batch processing, automated demonstrations and system testing.
- Multi-copy capability allows you to take full advantage of Apollo's multi-tasking environment to run multiple copies of HarPE simultaneously: no "temporary files" to be overwritten.
- Built-in log file maintains a complete record, including clock time, both of your input commands and messages from HarPE.

OSA SUPPORT

When you buy HarPE you will have one year of software support at no extra cost. This includes program upgrades and access to our highly trained technical support staff. To continue to receive these benefits after the first year, you may purchase an extended software support option.

PLATFORMS AND AVAILABILITY

HarPE is available on the Apollo platform. For further information please call or write.

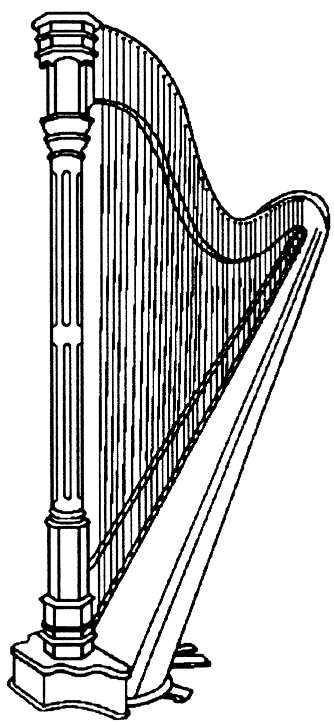
Optimization Systems Associates Inc.
P.O. Box 8083, Dundas, Ontario
Canada L9H 5E7

Tel (416) 628 8228, FAX (416) 628 8225

OSA

HarPE^{TM}

Technical Summary



HarPE™ Technical Summary

CONTENTS

INTRODUCTION 1

MEASUREMENT DATA 1

BUILT-IN DEVICE MODELS 1

CREATE YOUR OWN DEVICE MODELS
OR CUSTOMIZE LIBRARY MODELS 1

PARAMETER EXTRACTION FROM S-PARAMETERS 2

PARAMETER EXTRACTION FROM HARMONIC DATA 2

HarMONIZE LARGE- AND SMALL-SIGNAL SIMULATIONS! 2

THE WORLD'S MOST POWERFUL OPTIMIZERS 3

HarPE IS TRULY USER-FRIENDLY! 3

OSA SUPPORT 3

PLATFORMS AND AVAILABILITY 4

HarPE™ Technical Summary

INTRODUCTION

HarPE™ is dedicated to FET parameter extraction, device simulation and model building. Large- and small-signal data can be processed.

Released in conjunction with the 1989 IEEE MTT-S International Microwave Symposium, Long Beach, CA, HarPE is the world's first software system exploiting harmonic measurements for accurate nonlinear FET model parameter extraction. Furthermore, HarPE offers a comprehensive approach to FET device simulation.

HarPE emerges from several years of theoretical research and incorporates benchmark results by OSA [1-3]. Using HarPE, microwave circuit designers can make the most effective use of Libra, Touchstone, Microwave Harmonica and Super-Compact by supplying model parameters that truly belong to their own devices.

MEASUREMENT DATA

HarPE accepts device simulations or measurements in the form of:

- power spectra
- waveforms
- small-signal S-parameters
- DC bias data

HarPE features a flexible and intelligent input data format which is readily tailored by the user to accommodate raw data in different forms. HarPE can even be instructed to ignore keywords and data unknown to HarPE!

BUILT-IN DEVICE MODELS

Currently, HarPE supports three popular FET models:

- the Curtice and Ettenberg model
- the Materka and Kacprzak model
- the Raytheon (Statz *et al.*) model

All device models are implemented consistently with those in Libra and Microwave Harmonica. You can directly paste HarPE's output of FET parameters into the input files of Libra or Microwave Harmonica.

CREATE YOUR OWN DEVICE MODELS OR CUSTOMIZE LIBRARY MODELS

Model Customization

HarPE allows you to define your own device models using Fortran-like expressions.

Alternatively, you may customize or alter the built-in library models to reflect process, geometrical, physical or abstract functional dependence of any defined model parameters. Your customized models enjoy all computational options of HarPE.

In this manner, FET device dependence upon physical dimensions and material properties are at your fingertips. Model building is limited only by your imagination!

Typical Expressions

Any combinations of algebraic operations and standard mathematical functions are permitted, including:

```
EXP, LOG, LOG10, SQRT, ABS,
SIN, COS, TAN, ASIN, ACOS, ATAN,
SINH, COSH, TANH
```

A simple mechanism for evaluating conditional expressions is also provided.

Using these mathematical expressions, you can describe frequency dependence, temperature dependence, bias dependence, etc. For example, to describe a simple bias dependence of drain-source conductance GDS, you may use

```
GDS = GDSO + B * VG + C * VD
```

where VG and VD are gate and drain bias voltages and B and C are user-specified coefficients.

Model Creation

Create your own FET device model using HarPE!

For example, the drain current in the Curtice cubic model can be described as

```
V1 = VG_TAU*(1+BETA*(VDSO-VD_T));
CONDITION = POS(VD_T) + ZERO(VD_T);
   ! CONDITION = 1 if VD_T ≥ 0
   !           = 0 if VD_T < 0

DRAIN_CURRENT = CONDITION*(A0+A1*
                V1+A2*V1^2+A3*V1^3)*
                tanh(GAMMA* VD_T);
```

where VD_T is the instantaneous drain voltage at time T and VG_TAU is the instantaneous gate voltage delayed by TAU. A0, A1, A2, A3, VDS0, BETA and GAMMA are user-specified model parameters.

The HarPE Advantage

HarPE enables you to create models more simply and more effectively than you ever imagined possible! HarPE is the most versatile tool ever created for designing and gaining insight into proposed FET models.

Within about a day you should be able to create and test any new model. Consequently, HarPE is one of the most cost effective tools your company could invest in.

PARAMETER EXTRACTION FROM S-PARAMETERS

HarPE extracts large-signal model parameters from small-signal S-parameter data. HarPE simultaneously fits multi-bias, multi-frequency responses. The nonlinear device model automatically and smoothly links the model at different bias points.

PARAMETER EXTRACTION FROM HARMONIC DATA

As a unique feature, HarPE offers a truly nonlinear FET parameter extraction procedure which utilizes spectrum measurements, including DC bias information and power output at different harmonics measured under RF large-signal excitations.

The HarPE criterion for parameter extraction is that the computed and measured responses must simultaneously and optimally match at DC, fundamental frequency and higher harmonics. This approach directly exploits the nonlinear behavior of the device under test. The identifiability of a model is enhanced using a multi-circuit concept which accommodates multi-bias, multi-frequency and multi-input-level setups. The HarPE approach ensures the most reliable solutions of large-signal model parameters for subsequent circuit simulation with circuit simulators.

HarPE experiments have shown that all device parameters can be uniquely identified under actual high-frequency large-signal operating conditions, demonstrating the importance of higher harmonics in large-signal parameter extraction.

HarMONIZE LARGE- AND SMALL-SIGNAL SIMULATIONS!

HarPE works effectively as a device simulator with or without measurement data. HarPE provides graphical displays and numerical outputs and comparisons for a variety of device responses:

- DC IV curves
- harmonic responses vs. sweep of fundamental input powers
- harmonic responses vs. sweep of

fundamental excitation frequencies

- waveform responses of voltages and currents
- spectrum responses of powers, voltages and currents
- Smith chart and polar plots of S-parameters

HarPE seamlessly links large- and small-signal responses in a unified manner. The same circuit file and the same corresponding large-signal model description are used to compute S-parameter responses, DC IV curves, spectrum and waveform responses. For example, you can use a device model optimally extracted from S-parameter data to simulate spectrum and waveform responses. In addition, you can use HarPE to compare different device models.

THE WORLD'S MOST POWERFUL OPTIMIZERS

The numerical optimization of multiple nonlinear circuits is an extremely challenging task. That's why HarPE stands out as the world's first software boasting this capability. HarPE's secret lies in its implementation of state-of-the-art techniques, including OSA's nonlinear FAST™ adjoint sensitivity analysis and OSA's family of gradient optimizers. The 2-stage ℓ_1 optimizer has a proven track record in circuit optimization in general, and in device modeling in particular. The ℓ_2 optimizer implements a conventional least-squares objective and combines the power of the Levenberg-Marquardt and the quasi-Newton methods.

HarPE IS TRULY USER-FRIENDLY!

HarPE is so simple to use, you are immediately productive. You can quickly create models and explore them faster than you ever thought possible. HarPE may actually pay for itself in the first month of use!

The following features distinguish the most user-friendly CAD system ever offered to solve microwave simulation problems!

- Versatile, user-friendly input file.
- Built-in dual-window, dual-file full screen editor with search-replace, cut-paste, on-line macro, undo features and more.
- Fully integrated file manager, file parser and screen editor. The location of any error detected during file parsing is automatically highlighted by the editor.
- Logical, easy to understand, easy to use, menu-driven and window-driven commands with on-line help for every command.
- On-line User's Manual. You can add your own comments to the Manual, or copy syntax definitions and examples in the Manual and paste them directly into the circuit file, or define and search for arbitrary keywords (dynamic indexing).
- Macro file capability facilitates batch processing, automated demonstrations and system testing.
- Multi-copy capability allows you to take full advantage of Apollo's multi-tasking environment to run multiple copies of HarPE simultaneously: no "temporary files" to be overwritten.
- Built-in log file maintains a complete record, including clock time, both of your input commands and messages from HarPE.

OSA SUPPORT

When you buy HarPE from OSA, you are automatically entitled to one year of software support at no extra cost. This includes program upgrades and access to our highly trained technical support staff. To continue to receive these benefits after the first year, you may purchase an extended software support option.

PLATFORMS AND AVAILABILITY

HarPE is available now on the Apollo platform. For further information please call or write.

Optimization Systems Associates Inc.
P.O. Box 8083
Dundas, Ontario
Canada L9H 5E7

Tel (416) 628 8228
FAX (416) 628 8225

References

[1] J.W. Bandler, Q.J. Zhang, S. Ye and S.H. Chen, "Efficient large-signal FET parameter extraction using harmonics", *1989 IEEE MTT-S International Microwave Symposium Digest* (Long Beach, CA), June 1989.

[2] J.W. Bandler, S.H. Chen, S. Ye and Q.J. Zhang, "Integrated model parameter extraction using large-scale optimization concepts", *IEEE Trans. Microwave Theory Tech.*, vol. MTT-36, pp. 1629-1638, 1988.

[3] J.W. Bandler, Q.J. Zhang and R.M. Biernacki, "Practical, high speed gradient computation for harmonic balance simulators", *1989 IEEE MTT-S International Microwave Symposium Digest* (Long Beach, CA), June 1989.

HarPE™ Version 1.2 Technical Brief

INTRODUCTION

HarPE™ is dedicated to device parameter extraction, device simulation and model building. Large- and small-signal data can be processed.

HarPE is the world's first software system exploiting harmonic data for accurate nonlinear model parameter extraction, while offering a comprehensive approach to device simulation.

Using HarPE, circuit designers can make the most effective use of Libra, Touchstone, Microwave Harmonica and Super-Compact by supplying model parameters that truly belong to their own devices.

MEASUREMENT DATA

HarPE accepts device measurements in the form of any combination of:

- power spectra
- waveforms
- small-signal S-parameters
- DC bias data

HarPE features a flexible and intelligent input data format which is readily tailored by the user to accommodate raw data in different forms.

BUILT-IN DEVICE MODELS

HarPE supports the following device models

- the Curtice and Ettenberg FET model
- the Materka and Kacprzak FET model
- the Raytheon (Statz *et al.*) FET model
- the Gummel and Poon model of bipolar NPN and PNP transistors

Device models are implemented consistently with Libra and Microwave Harmonica, into whose input files you can directly paste HarPE's output of device parameters.

MEASUREMENT AND SIMULATION ENVIRONMENT

You can use one of the predefined extrinsic linear subcircuits for extremely efficient linear simulations. Alternatively, you can define your own parasitic subcircuit using built-in linear elements or supplying externally obtained measurement or simulation data.

HarPE accepts general complex, frequency dependent and user-defined terminations to provide for maximum flexibility for the simulation and parameter extraction environment you need.

CREATE OR CUSTOMIZE MODELS

You may customize or alter the built-in library models to reflect any functional dependence of any defined model parameters. Your customized models will enjoy all computational options of HarPE.

Combinations of algebraic operations and standard functions are permitted, including:

```
EXP, LOG, LOG10, SQRT, ABS,
SIN, COS, TAN, ASIN, ACOS, ATAN,
SINH, COSH, TANH
```

A simple mechanism for evaluating conditional expressions is also provided.

Using these mathematical expressions, you can describe frequency dependence, temperature or bias dependence, etc. For example, to describe a simple bias dependence of drain-source conductance GDS, you may use

```
GDS = GDS0 + B * VG + C * VD
```

where VG and VD are gate and drain bias voltages and B and C are user-specified coefficients.

HarPE allows you to define your own device models using Fortran-like expressions.

HarPE enables you to create models more simply and more effectively than you ever imagined possible! HarPE is the most versatile tool ever created for designing and gaining insight into proposed device models.

Within about a day you should be able to create and test any new model. Consequently, HarPE is one of the most cost effective tools your company could invest in.

PARAMETER EXTRACTION FROM S-PARAMETERS

HarPE extracts large-signal model parameters from small-signal S-parameter and/or DC measurement data. HarPE simultaneously fits multi-bias, multi-frequency responses. The nonlinear characteristics automatically and smoothly links the model at different bias points. In addition, HarPE can directly optimize the model for the best match of such functions of S-parameters as stability factor and maximum available gain.

EXTRACTION FROM HARMONIC DATA

As a unique feature, HarPE offers a truly nonlinear parameter extraction procedure which utilizes spectrum measurements, including DC bias information and power output at different harmonics measured under RF large-signal excitations.

The HarPE criterion for parameter extraction is that the computed and measured responses must simultaneously and optimally match at DC, fundamental frequency and higher harmonics. The HarPE approach ensures the most reliable solutions of large-signal model parameters for subsequent circuit simulation with circuit simulators.

PARAMETER EXTRACTION FROM BOTH S-PARAMETERS AND HARMONIC DATA

HarPE allows you to take advantage of all possible measurement data you may have. Any combination of large-signal harmonic data, small-signal S-parameter data and/or DC measurement data can be simultaneously processed by HarPE's optimizers. The nonlinear device model automatically and smoothly links the model for all different measurement conditions.

HarMONIZE LARGE- AND SMALL-SIGNAL SIMULATIONS!

HarPE works effectively as a device simulator with or without measurement data. HarPE provides graphical displays and numerical outputs and comparisons for

- DC IV curves
- harmonic responses vs. sweep of fundamental input powers
- harmonic responses vs. sweep of fundamental excitation frequencies
- voltage and current waveforms
- power, voltage and current spectra
- Smith chart and polar plots of S-parameters
- stability factor and maximum available gain
- arbitrary user-defined response functions

HarPE seamlessly links large- and small-signal responses in a unified manner. The same circuit file and the same corresponding large-signal model description are used to compute S-parameter responses, DC IV curves, spectrum and waveform responses. For example, you can use a device model optimally extracted from S-parameter data to simulate spectrum and waveform responses.

HarPE IS TRULY USER-FRIENDLY!

HarPE is so simple to use, you are immediately productive. You can quickly create models and explore them faster than you ever thought possible. HarPE may actually pay for itself in the first month of use!

OSA SUPPORT

When you buy HarPE you will have one year of software support at no extra cost. This includes program upgrades and access to our highly trained technical support staff. To continue to receive these benefits after the first year, you may purchase an extended software support option.

PLATFORMS AND AVAILABILITY

HarPE is available on Apollo and Hewlett-Packard workstations. For further information please call or write.

Optimization Systems Associates Inc.
P.O. Box 8083, Dundas, Ontario
Canada L9H 5E7

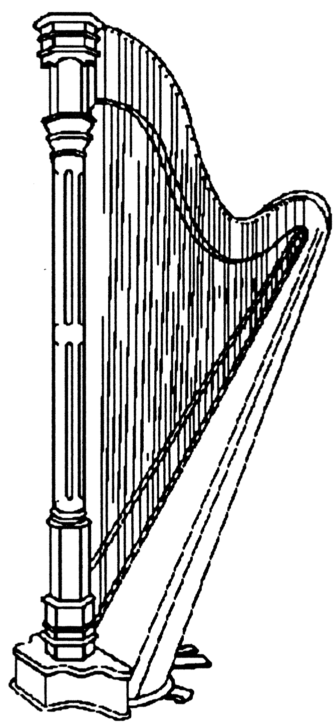
Tel (416) 628 8228
FAX (416) 628 8225

OSA

HarPE™

Version 1.2

Technical Summary



HarPE™ Version 1.2 Technical Summary

CONTENTS

INTRODUCTION 1

MEASUREMENT DATA 1

BUILT-IN DEVICE MODELS 1

MEASUREMENT AND SIMULATION ENVIRONMENT 1

CREATE YOUR OWN DEVICE MODELS
OR CUSTOMIZE LIBRARY MODELS 1

PARAMETER EXTRACTION FROM S-PARAMETERS 2

PARAMETER EXTRACTION FROM HARMONIC DATA 2

PARAMETER EXTRACTION FROM BOTH
S-PARAMETERS AND HARMONIC DATA 3

HarMONIZE LARGE- AND SMALL-SIGNAL SIMULATIONS! 3

THE WORLD'S MOST POWERFUL OPTIMIZERS 3

HarPE IS TRULY USER-FRIENDLY! 3

OSA SUPPORT 4

VERSION 1.2 ENHANCEMENTS 4

PLATFORMS AND AVAILABILITY 4

HarPE™ Version 1.2 Technical Summary

INTRODUCTION

HarPE™ is dedicated to device parameter extraction, device simulation and model building. Large- and small-signal data can be processed.

Released in conjunction with the 1989 IEEE MTT-S International Microwave Symposium, Long Beach, CA, HarPE is the world's first software system exploiting harmonic measurements for accurate nonlinear model parameter extraction. Furthermore, HarPE offers a comprehensive approach to device simulation.

HarPE emerges from several years of theoretical research and incorporates benchmark results by OSA [1-3]. Using HarPE, circuit designers can make the most effective use of Libra, Touchstone, Microwave Harmonica and Super-Compact by supplying model parameters that truly belong to their own devices.

MEASUREMENT DATA

HarPE accepts device measurements in the form of any combination of:

- power spectra
- waveforms
- small-signal S-parameters
- DC bias data

HarPE features a flexible and intelligent input data format which is readily tailored by the user to accommodate raw data in different forms. HarPE can even be instructed to ignore keywords and data unknown to HarPE!

BUILT-IN DEVICE MODELS

HarPE supports the following device models

- the Curtice and Ettenberg FET model
- the Materka and Kacprzak FET model
- the Raytheon (Statz *et al.*) FET model
- the Gummel and Poon model of bipolar NPN and PNP transistors

All device models are implemented consistently with those in Libra and Microwave Harmonica. You can directly paste HarPE's output of device parameters into the input files of Libra or Microwave Harmonica.

MEASUREMENT AND SIMULATION ENVIRONMENT

You can use one of the predefined extrinsic linear subcircuits for extremely efficient linear simulations. Alternatively, you can define your own parasitic subcircuit using built-in linear elements or supplying externally obtained measurement or simulation data.

HarPE accepts general complex, frequency dependent and user-defined terminations to provide for maximum flexibility for the simulation and parameter extraction environment you need.

CREATE YOUR OWN DEVICE MODELS OR CUSTOMIZE LIBRARY MODELS

Model Customization

You may customize the built-in library models to reflect process, geometrical, physical or abstract functional dependence of any defined model parameter. Your customized models will enjoy all computational options of HarPE.

In this manner, device dependence upon physical dimensions and material properties are at your fingertips.

Typical Expressions

Any combinations of algebraic operations and standard mathematical functions are permitted, including:

```
EXP, LOG, LOG10, SQRT, ABS,
SIN, COS, TAN, ASIN, ACOS, ATAN,
SINH, COSH, TANH
```

A simple mechanism for evaluating conditional expressions is also provided.

Using these mathematical expressions, you can describe frequency dependence, temperature or bias dependence, etc. For example, to describe a simple bias dependence of drain-source conductance GDS, you may use

```
GDS = GDS0 + B * VG + C * VD
```

where VG and VD are gate and drain bias voltages and B and C are user-specified coefficients.

Model Creation

Create your own device model using HarPE!

HarPE allows you to define your own device models using Fortran-like expressions. For example, the drain current in the Curtice cubic FET model can be described as

```
V1 = VG_TAU*(1+BETA*(VDS0-VD_T));
CONDITION = POS(VD_T) + ZERO(VD_T);
  ! CONDITION = 1 if VD_T ≥ 0
  !           = 0 if VD_T < 0

DRAIN_CURRENT = CONDITION*(A0+A1*
                V1+A2*V1^2+A3*V1^3)*
                tanh(GAMMA* VD_T);
```

where VD_T is the instantaneous drain voltage at time T and VG_TAU is the instantaneous gate voltage delayed by TAU. A0, A1, A2, A3, VDS0, BETA and GAMMA are user-specified model parameters.

The HarPE Advantage

HarPE enables you to create models more simply and more effectively than you ever imagined possible! HarPE is the most versatile tool ever created for designing and gaining insight into proposed models.

Within about a day you should be able to create and test any new model. Consequently, HarPE is one of the most cost effective tools your company could invest in.

PARAMETER EXTRACTION FROM S-PARAMETERS

HarPE extracts large-signal model parameters from small-signal S-parameter data. HarPE simultaneously fits multi-bias, multi-frequency responses. The nonlinear device model automatically and smoothly links the model at different bias points. In addition, HarPE can directly optimize the model for the best match of such functions of S-parameters as stability factor and maximum available gain.

PARAMETER EXTRACTION FROM HARMONIC DATA

As a unique feature, HarPE offers a truly nonlinear device parameter extraction procedure which utilizes spectrum measurements, including DC bias information and power output at different harmonics measured under RF large-signal excitations.

The HarPE criterion for parameter extraction is that the computed and measured responses must simultaneously and optimally match at DC, fundamental frequency and higher harmonics. This approach directly exploits the nonlinear behavior of the device under test. The identifiability of a model is enhanced using a multi-circuit concept which accommodates multi-bias, multi-frequency and multi-input-level setups. The HarPE approach ensures the most reliable solutions of large-signal model parameters for subsequent circuit simulation with circuit simulators.

HarPE experiments have shown that all device parameters can be uniquely identified under actual high-frequency large-signal operating conditions, demonstrating the importance of higher harmonics in large-signal parameter extraction.

PARAMETER EXTRACTION FROM BOTH S-PARAMETERS AND HARMONIC DATA

HarPE allows you to take advantage of all possible measurement data you may have. Any combination of large-signal harmonic data, small-signal S-parameter data and/or DC measurement data can be simultaneously processed by HarPE's optimizers. The nonlinear device model automatically and smoothly links the model for all different measurement conditions.

HarMONIZE LARGE- AND SMALL-SIGNAL SIMULATIONS!

HarPE works effectively as a device simulator with or without measurement data. HarPE provides graphical displays and numerical outputs and comparisons for a variety of device responses:

- DC IV curves
- harmonic responses vs. sweep of fundamental input powers
- harmonic responses vs. sweep of fundamental excitation frequencies
- voltage and current waveforms
- power, voltage and current spectra
- Smith chart and polar plots of S-parameters
- stability factor and maximum available gain
- arbitrary user-defined response functions

HarPE seamlessly links large- and small-signal responses in a unified manner. The same circuit file and the same corresponding large-signal model description are used to compute S-parameter responses, DC IV curves, spectrum and waveform responses. For example, you can use a device model optimally extracted from S-parameter data to simulate spectrum and waveform responses.

HarPE offers postprocessing of the simulation results for arbitrary user-defined response functions. You may use any combination of algebraic operations and standard mathematical functions to manipulate the simulated large- and small-signal responses in order to create your own response functions for graphical display and numerical output.

In addition, you can use HarPE to compare different device models.

THE WORLD'S MOST POWERFUL OPTIMIZERS

The numerical optimization of multiple nonlinear circuits is an extremely challenging task. That's why HarPE stands out as the world's first software boasting this capability. HarPE's secret lies in its implementation of state-of-the-art techniques, including OSA's nonlinear FAST™ adjoint sensitivity analysis and OSA's family of gradient optimizers. The 2-stage ℓ_1 optimizer has a proven track record in circuit optimization in general, and in device modeling in particular. The ℓ_2 optimizer implements a conventional least-squares objective and combines the power of the Levenberg-Marquardt and the quasi-Newton methods.

HarPE IS TRULY USER-FRIENDLY!

HarPE is so simple to use, you are immediately productive. You can quickly create models and explore them faster than you ever thought possible. HarPE may actually pay for itself in the first month of use!

The following features distinguish the most user-friendly CAD system ever offered to circuit designers!

- Versatile, user-friendly input file.
- Built-in dual-window, dual-file full screen editor with search-replace, cut-paste, on-line macro, undo features and more.
- Fully integrated file manager, file parser and screen editor. The location of any error detected during file parsing is automatically highlighted by the editor.

- Logical, easy to understand, easy to use, menu-driven and window-driven commands with on-line help for every command.

- On-line User's Manual. You can add your own comments to the Manual, or copy syntax definitions and examples in the Manual and paste them directly into the circuit file, or define and search for arbitrary keywords (dynamic indexing).

- Macro file capability facilitates batch processing, automated demonstrations and system testing.

- Multi-copy capability allows you to take full advantage of a multi-tasking environment to run multiple copies of HarPE simultaneously: no "temporary files" to be overwritten.

- Built-in log file maintains a complete record, including clock time, both of your input commands and messages from HarPE.

OSA SUPPORT

When you buy HarPE from OSA, you are automatically entitled to one year of software support at no extra cost. This includes program upgrades and access to our highly trained technical support staff. To continue to receive these benefits after the first year, you may purchase an extended software support option.

VERSION 1.2 ENHANCEMENTS

The following new features distinguish HarPE Version 1.2 from HarPE Version 1.1

- built-in models of bipolar transistors

- expanded linear element library

- capability of handling arbitrary topology allowing you to define your own linear environment

- general complex, frequency dependent and user-defined terminations

- user-defined reference resistance for S-parameters

- built-in calculations of the stability factor and maximum available gain for both output and parameter extraction

- parameter extraction from simultaneously processed small- and large-signal data

- postprocessing capability allowing you to define arbitrary response functions for both graphical and numerical output

PLATFORMS AND AVAILABILITY

HarPE Version 1.2 is available now on Apollo and Hewlett-Packard workstations. For further information please call or write.

Optimization Systems Associates Inc.
P.O. Box 8083
Dundas, Ontario
Canada L9H 5E7

Tel (416) 628 8228 FAX (416) 628 8225

References

[1] J.W. Bandler, Q.J. Zhang, S. Ye and S.H. Chen, "Efficient large-signal FET parameter extraction using harmonics", *1989 IEEE MTT-S International Microwave Symposium Digest* (Long Beach, CA, 1989), pp. 577-580.

[2] J.W. Bandler, S.H. Chen, S. Ye and Q.J. Zhang, "Integrated model parameter extraction using large-scale optimization concepts", *IEEE Trans. Microwave Theory Tech.*, vol. MTT-36, 1988, pp. 1629-1638.

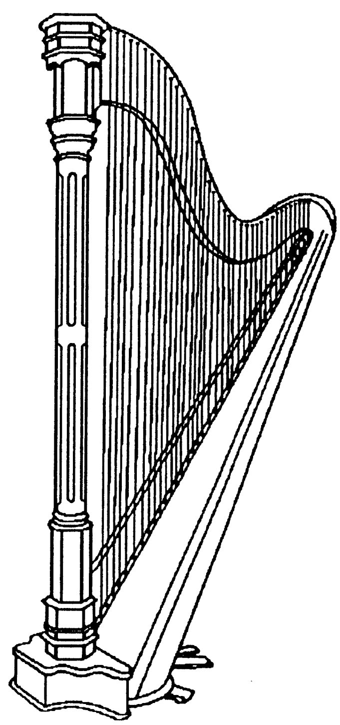
[3] J.W. Bandler, Q.J. Zhang and R.M. Biernacki, "Practical, high speed gradient computation for harmonic balance simulators", *1989 IEEE MTT-S International Microwave Symposium Digest* (Long Beach, CA, 1989), pp. 363-366.

OSA

HarPE™

Version 1.3

Technical Summary



HarPE™ Version 1.3 Technical Summary

CONTENTS

INTRODUCTION 1

DEVICE SIMULATION 1

CIRCUIT DESCRIPTION 2

OPTIMIZATION 2

PARAMETER EXTRACTION 2

CUSTOMIZING MODELS AND RESPONSE FUNCTIONS 3

USER INTERFACE 4

COMPATIBILITY WITH OTHER CAD SOFTWARE 4

OSA SUPPORT 4

PLATFORMS AND AVAILABILITY 4

HarPE™ Version 1.3 Technical Summary

INTRODUCTION

HarPE™ has emerged from many years of theoretical research and incorporates benchmark results by Optimization Systems Associates Inc.

HarPE Version 1.3 is an advanced, powerful and user-friendly CAD system dedicated to device nonlinear simulation, characterization and optimization. HarPE is the world's first software system for complete device characterization using harmonic measurements.

DEVICE SIMULATION

HarPE offers a comprehensive approach to device-oriented, seamlessly integrated large-signal, small-signal and DC simulations using a unified circuit description. When needed, the small-signal parameters are automatically linearized from the nonlinear model.

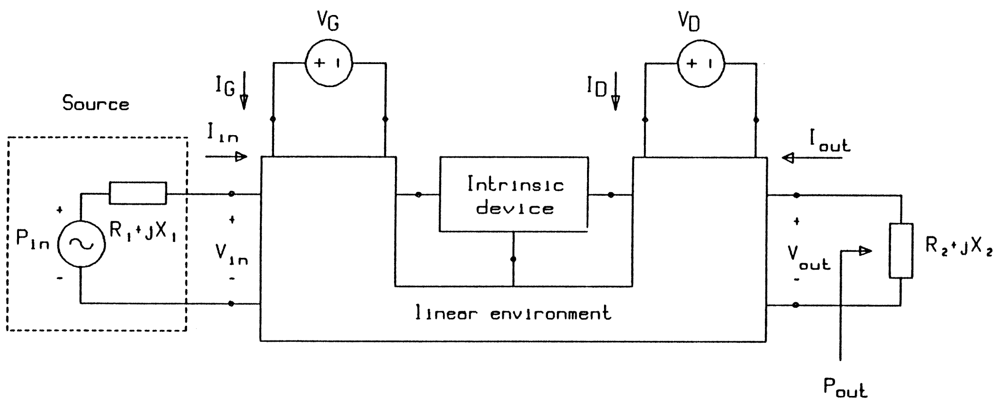
The overall schematic of a single-device circuit is shown below. P_{in} is the RF source described by its available power, R_1+jX_1 is the source impedance, P_{out} is the actual AC power delivered to the load R_2+jX_2, V_G represents the gate (base) DC bias voltage, and V_D the drain (collector) DC bias voltage.

At AC, HarPE calculates the input and output voltages and currents V_{in}, V_{out}, I_{in} and I_{out}, output power P_{out}, and small-signal S-parameters. At DC, HarPE calculates the currents at the bias ports, namely I_G and I_D.

For harmonic balance simulation, single-tone excitation is considered. Arbitrary combinations of bias sweep, frequency sweep and power sweep are accommodated.

Built-in responses for high-quality graphical display and numerical output include:

- frequency spectra of powers, voltages and currents
- time-domain waveforms of voltages and currents
- spectrum responses vs. sweep of available input power
- spectrum responses vs. sweep of fundamental excitation frequency
- DC IV curves
- small-signal parameters linearized from nonlinear models
- Smith chart and polar plots of S-parameters
- maximum available gain and stability factor



Schematic representation of a single-device circuit

CIRCUIT DESCRIPTION

To represent the intrinsic part of the circuit, you can use one of the built-in nonlinear intrinsic models to achieve maximum computational efficiency. Currently, the nonlinear intrinsic device model library includes:

- the Curtice and Ettenberg FET model
- the Materka and Kacprzak FET model
- the Raytheon (Statz *et al.*) FET model
- the Gummel and Poon model of bipolar NPN and PNP transistors

Alternatively, you can create custom device models using HarPE's unique Expression Block.

To represent the linear environment you can use one of the predefined extrinsic super-components for extremely efficient linear simulations. Alternatively, you can define your own linear environment using built-in linear elements or supplying externally obtained measurement or simulation data. Currently, the linear library includes four extrinsic super-components, lumped resistors, capacitors and inductors, as well as two transmission line models.

HarPE accepts general complex, frequency dependent and user-defined terminations to provide for maximum flexibility for the simulation environment you need.

OPTIMIZATION

The power of HarPE lies in its implementation of state-of-the-art techniques. The gradient-based minimax, ℓ_1 and least-squares optimizers have a proven track record in circuit optimization.

Combining the optimizers with OSA's unique nonlinear FAST™ adjoint sensitivity analysis, HarPE provides solutions to your design and modeling problems with unparalleled speed, accuracy, consistency and robustness.

The functions, goals and measured data for optimization can be any combination of power spectra, waveforms, S-parameters and DC IV data. Stability factor and maximum available gain can also be optimized.

In addition to built-in responses, arbitrary user-defined functions can be optimized. Even the goals and window specifications can be user-defined expressions. This capability makes HarPE a flexible system for your modeling and design optimization needs.

PARAMETER EXTRACTION

Employing a unique multi-circuit concept, HarPE processes simultaneously multi-bias, multi-frequency, and multi-input-power-level measurements. This approach increases model identifiability and ensures a reliable and consistent solution suitable for large-signal, small-signal and DC simulation. In spite of the fact that numerical optimization of multiple nonlinear circuits is an extremely challenging task, HarPE stands out as the world's first software boasting this capability.

Parameter Extraction from Harmonic Data

HarPE is the world's first software system for parameter extraction from harmonic data. It offers a truly nonlinear device parameter extraction procedure which utilizes spectrum measurements, including DC bias information and power output at different harmonics measured under RF large-signal excitations.

The HarPE criterion for parameter extraction is that the computed and measured responses must simultaneously and optimally match at DC, fundamental frequency and higher harmonics. This approach directly exploits the nonlinear behavior of the device under test and ensures the most reliable solutions of large-signal model parameters for subsequent circuit simulation with circuit simulators.

HarPE experiments have shown that all device parameters can be uniquely identified under actual high-frequency large-signal operating conditions, demonstrating the importance of higher harmonics in large-signal parameter extraction.

Parameter Extraction from S-parameters and/or DC Data

HarPE can extract the nonlinear model parameters from small-signal S-parameter data or even just from DC data.

In fact, HarPE accepts device measurements in the form of any combination of power spectra, waveforms, small-signal S-parameters and/or DC data. The nonlinear device model automatically and smoothly links the model at all available bias points.

CUSTOMIZING MODELS AND RESPONSE FUNCTIONS

Using Fortran-like expressions, you can create your own device models or modify the built-in models to define process, geometrical, physical or abstract functional dependence of any model parameters. You can also define your own response functions, as well as goals and specifications for optimization. All these are achieved within HarPE's unique Expression Block. No subroutines need to be written, compiled or linked. This allows you to test, verify and even optimize in-house models effortlessly.

User-defined models and response functions enjoy the power of HarPE in simulation and optimization just like the built-in models and responses.

Typical Expressions

Any combinations of algebraic operations and standard mathematical functions are permitted in formulating an expression, including:

```
EXP, LOG, LOG10, SQRT, ABS,
SIN, COS, TAN, ASIN, ACOS, ATAN,
SINH, COSH, TANH
```

Conditional expressions are also supported. A set of predefined labels allows you to refer to the program's internal variables and built-in responses, such as the frequency, bias voltages, instantaneous intrinsic voltages, input power, output power spectrum components, DC current and S-parameters.

For instance, you may use the expression

```
GDS = GDS0 + B * VG + C * VD
```

to describe the bias dependence of a FET drain-source conductance GDS, where VG and VD are predefined labels for gate and drain bias voltages and B and C are your own model parameters.

Define Your Own Model Equations

As an example of a user-defined model equation, the drain current in the Curtice cubic FET model can be directly defined in the input file as

```
V1 = VGS_TAU*(1+BETA*(VDS0-VDS_T));
CONDITION = POS(VDS_T) + ZERO(VDS_T);
   ! CONDITION = 1 if VDS_T ≥ 0
   !             0 if VDS_T < 0

DRAIN_CURRENT = CONDITION *
        (A0+A1*V1+A2*V1^2+A3*V1^3) *
        tanh(GAMMA*VDS_T);
```

where VDS_T is the instantaneous drain voltage at time T and VGS_TAU is the instantaneous gate voltage delayed by TAU. A0, A1, A2, A3, VDS0, BETA and GAMMA are user-defined model parameters.

Define Your Own Responses

HarPE offers postprocessing of the simulation results for arbitrary user-defined response functions. You may use any combination of algebraic operations and standard mathematical functions to manipulate the simulated large- and small-signal responses in order to create your own response functions for graphical display, numerical output and optimization.

For example, you may define a custom response function such as

```
POWER_RATIO_DBM = POUT1 - PIN;
```

where PIN and POUT1 are predefined labels for the input power and the output power at fundamental frequency in dBm, respectively.

USER INTERFACE

The sophisticated capabilities of HarPE are presented through a polished and friendly user-interface. It is menu-driven, supports a mouse and features an integrated full screen editor, clear syntax, flexible data format, high-quality graphics, on-line help and on-line User's Manual. The following highlight some of the features.

- Logically designed menu-driven and window-driven commands with on-line help for every command.
- Mouse support facilitates quick and fluent operation. However, the program is fully operational from a keyboard without a mouse.
- Clear and uncomplicated input file syntax with flexible and user-designable data format.
- Built-in dual-window, dual-file full screen editor with search and replace, cut and paste, buffered undo, macros, and more.
- File parser fully integrated into the screen editor. The location of any syntax error is automatically highlighted on screen by the editor.
- On-line User's Manual. You can add your own comments to the Manual, or copy syntax definitions and examples in the Manual and paste them directly into your circuit file, or define and search for arbitrary keywords (dynamic indexing).
- The program can be operated in three different modes: run interactively, driven by a macro file, or run as a silent background process.
- Automatically generated log file maintains a complete record of the operations, including clock time, user commands and messages.

COMPATIBILITY WITH OTHER CAD SOFTWARE

HarPE can help you make the most effective use of other CAD products such as Libra™/Touchstone® or Microwave Harmonica® /Super-Compact® by supplying model parameters that truly belong to your own devices. The built-in nonlinear device models can be directly exported into Libra and Microwave Harmonica input files.

HarPE also supports linear subcircuits described by "black box" S, Y, or Z data imported from external simulators.

In addition to its flexible data format, HarPE directly accepts measured data from Cascade Microtech's MicroCAT™ Test Executive system and data in the MDIF (Measurement Data Interchange Format) format.

OSA SUPPORT

As a user of HarPE, you can rely on the expedient and professional support from OSA's technical experts, including the authors of the program.

With your acquisition of HarPE you are automatically entitled to a standard 90 day warranty. An option of free software upgrades and maintenance for one year is also available.

PLATFORMS AND AVAILABILITY

HarPE Version 1.3 is available on Apollo, Hewlett-Packard and SUN workstations. For further information please call or write.

Optimization Systems Associates Inc.
P.O. Box 8083
Dundas, Ontario
Canada L9H 5E7

Tel 416 628 8228 Fax 416 628 8225

HarPE™ Version 1.3 Highlights

- Accurate characterization from large-signal, small-signal and DC measurements
- **Design optimization with user-defined responses and goals using expressions, including window specifications**
- L1, least-squares and **minimax** optimizers
- A library of built-in device models and linear elements, including new **extrinsic super components** and **transmission lines**
- User-defined device models using expressions
- User-defined response functions using expressions
- Fast, reliable large- and small-signal device simulation
- Graphical display of spectra, waveforms, power and frequency sweeps, DC IV curves, **small-signal model parameters**, Smith chart, polar plot, stability factor and maximum available gain, and user-defined responses
- Menu-driven, window-driven, **mouse-driven**
- Integrated full screen editor
- **Direct data import from Cascade Microtech measurement system**
- Many more features described in a Technical Summary available upon request

Features in **bold** are new to Version 1.3.

Optimization Systems Associates Inc.
P.O. Box 8083, Dundas, Ontario, Canada L9H 5E7
Telephone 416 628 8228, Fax 416 628 8225

HarPE™ Versions 1.4/1.4+S Technical Brief

INTRODUCTION

HarPE™ Version 1.4 is an advanced, powerful, integrated CAD system dedicated to device nonlinear characterization, simulation and optimization. HarPE™ Version 1.4+S additionally contains an optional statistical modeling and analysis module.

DEVICE SIMULATION

HarPE offers a comprehensive approach to device-oriented, seamlessly integrated large-signal, small-signal and DC simulations using a unified circuit description. When needed, the small-signal parameters are automatically linearized from the nonlinear model.

The overall HarPE circuit topology consists of an intrinsic nonlinear device encompassed by a linear environment to which are connected the RF and bias sources and the load. 1-port and 2-port configurations are accommodated.

At AC, HarPE calculates the RF input and output voltages and currents, output power, and small-signal S-parameters. At DC, HarPE calculates the currents at the bias ports.

Arbitrary combinations of bias, frequency and power sweeps are available. Parameter sweeps can also be simulated.

Built-in responses include:

- power, voltage and current spectra
- voltage and current waveforms
- DC IV curves
- small-signal parameters linearized from nonlinear models
- Smith chart and polar plots of S-parameters
- maximum available gain and stability factor

CIRCUIT DESCRIPTION

To represent the intrinsic part of the circuit, you can use one of the built-in nonlinear intrinsic device models:

- the semiconductor diode model
- the Curtice and Ettenberg FET model
- the Materka and Kacprzak FET model
- the Raytheon (Statz *et al.*) FET model
- the Khatibzadeh and Trew FET model
- the Gummel and Poon model of bipolar NPN and PNP transistors

Alternatively, you can create custom device models using HarPE's unique Expression Block.

To represent the linear environment you can use one of the predefined extrinsic super-components for extremely efficient linear simulations. Alternatively, you can define your own linear environment using built-in linear elements or supplying externally obtained measurement or simulation data.

HarPE accepts general complex, frequency dependent and user-defined terminations to provide for maximum flexibility for the simulation environment you need.

OPTIMIZATION

The power of HarPE lies in its implementation of state-of-the-art techniques. The gradient-based minimax, ℓ_1 and least-squares optimizers have a proven track record in circuit optimization.

Combining the optimizers with OSA's unique nonlinear FAST™ adjoint sensitivity analysis, HarPE provides solutions to your design and modeling problems with unparalleled speed, accuracy, consistency and robustness.

A distinct feature of sensitivity displays is available to help you to select the most crucial parameters for optimization while fixing those that have lesser influence.

In addition to built-in responses, arbitrary user-defined functions can be optimized. Even the goals and window specifications can be user-defined expressions. This capability makes HarPE a flexible system for your modeling and design optimization needs.

PARAMETER EXTRACTION

HarPE is the world's first software system for parameter extraction from harmonic data. It offers a truly nonlinear device parameter extraction procedure which utilizes spectrum measurements, including DC bias information and power output at different harmonics measured under RF large-signal excitations.

HarPE can also extract the nonlinear model parameters from small-signal S-parameter data or even just from DC data. In fact, HarPE accepts device measurements in the form of any combination of power spectra, waveforms, small-signal S-parameters and/or DC data. The nonlinear device model automatically and smoothly links the model at all available bias, frequency and input-power-level points.

CUSTOMIZING MODELS AND RESPONSE FUNCTIONS

Using Fortran-like expressions, you can create your own device models or modify the built-in models. You can also define your own response functions, as well as goals and specifications for optimization. All these are achieved within HarPE's unique Expression Block. No subroutines need to be written, compiled or linked.

User-defined models and response functions enjoy the power of HarPE in simulation and optimization just like the built-in models and responses.

Combinations of algebraic operations and standard mathematical functions are permitted in formulating an expression. Conditional expressions are also supported.

A set of predefined labels allows you to refer to the program's internal variables and built-in responses, such as the frequency, bias voltages, instantaneous intrinsic voltages, input power, output power spectrum components, DC current and S-parameters.

USER INTERFACE

The sophisticated capabilities of HarPE are presented through a polished and friendly user-interface. It is menu-driven, supports a mouse and features an integrated full screen editor, clear syntax, flexible and user-designable data format, high-quality graphics, on-line help and on-line User's Manual.

COMPATIBILITY WITH OTHER CAD SOFTWARE

HarPE can help you make the most effective use of other CAD products by supplying model parameters that truly belong to your own devices.

HarPE supports linear subcircuits described by "black box" S, Y, or Z data imported from external simulators.

In addition to its flexible data format, HarPE directly accepts measured data from Cascade Microtech's MicroCAT™ Test Executive system and data in the MDIF (Measurement Data Interchange Format) format.

STATISTICAL MODELING AND ANALYSIS

An optional add-on module for statistical modeling and analysis is available in HarPE Version 1.4+S.

Any of the built-in or user defined models can be used for statistical modeling. Based on a large set of measurement data from multiple devices, a multi-device parameter extraction procedure provides a sample of device models. This can be further postprocessed to create a consolidated statistical model, immediately ready for statistical Monte Carlo analysis.

OSA SUPPORT

With your acquisition of HarPE you are entitled to a standard 90 day warranty.

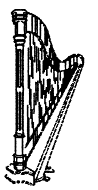
An option of free software program upgrades and maintenance for one year is also available. To continue to receive these benefits after the first year you may purchase an extended software support option.

PLATFORMS AND AVAILABILITY

HarPE Versions 1.4 and 1.4+S are available on Apollo, Hewlett-Packard and Sun workstations. For further information please call or write.

Optimization Systems Associates Inc.
P.O. Box 8083, Dundas, Ontario
Canada L9H 5E7

Tel 416 628 8228   Fax 416 628 8225



HarPE™ Version 1.4+S Statistical Option

INTRODUCTION

Advanced features of statistical modeling and analysis are now available as an add-on module for HarPE™. The package is released as HarPE Version 1.4+S.

Any of the built-in or user defined models can be used for statistical modeling and analysis. HarPE is the first commercial software system to offer a physics based FET model *together with* statistical capabilities.

MULTI-DEVICE MODELING

HarPE Version 1.4+S implements a multi-device optimization and postprocessing approach to statistical modeling. Based on a large set of measurement data from multiple devices (large-signal power spectra, small-signal S-parameters, and/or DC IV data), a multi-device parameter extraction procedure provides a sample of device models. OSA's gradient-based ℓ_1 optimizer provides a fast and reliable tool for the underlying parameter extraction.

POSTPROCESSING

Histograms showing parameter spreads and scatter diagrams showing parameter correlations are displayed. The sample of device models can be further postprocessed to generate a consolidated statistical model, which is then back annotated to produce a HarPE circuit file immediately suitable for Monte Carlo analysis.

Parameter correlations are included in the model. For parameters exhibiting substantially non-Gaussian distributions, the user can keep the histograms as discrete approximations of the marginal density functions.

PROCESSING RAW DATA

The statistical postprocessor in HarPE Version 1.4+S can also be applied to arbitrary raw data supplied by the user. The statistics (means, standard deviations and correlations) of such data can be analyzed, and histograms and scatter diagrams can be displayed.

MONTE CARLO ANALYSIS

HarPE Version 1.4+S features statistical Monte Carlo analysis including large-signal, DC and small-signal simulations. Uniform, normal, exponential and lognormal distributions with absolute or relative tolerances can be used. Histograms, run charts, and sweep diagrams are displayed. Yield can be computed from specifications on spectra, S-parameters, DC currents and arbitrary user-defined functions. The statistical responses can be saved as numerical data files. The back annotated circuit file produced by the statistical modeling option can be directly used for Monte Carlo analysis.

COMPATIBILITY WITH OTHER CAD SOFTWARE

In addition to its flexible data format, HarPE Version 1.4+S directly accepts multi-device measurement data from Cascade Microtech's MicroCAT™ Test Executive system.

OSA SUPPORT

HarPE version 1.4+S includes free software upgrades and maintenance for one year. To continue to receive these benefits after the first year you may purchase an extended software support option.

PLATFORMS AND AVAILABILITY

HarPE Version 1.4+S is available on Apollo, Hewlett-Packard and Sun workstations. For further information please call or write.

Optimization Systems Associates Inc.
P.O. Box 8083, Dundas, Ontario
Canada L9H 5E7

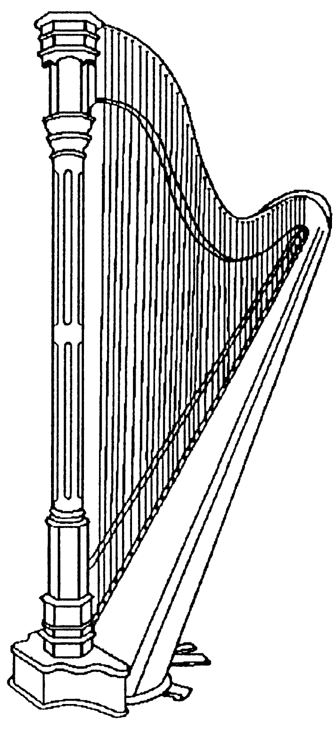
Tel 416 628 8228   Fax 416 628 8225

OSA

HarPE™

Versions 1.4/1.4+S

Technical Summary



HarPE™ Versions 1.4/1.4+S Technical Summary

CONTENTS

INTRODUCTION 1

DEVICE SIMULATION 1

CIRCUIT DESCRIPTION 2

OPTIMIZATION 2

PARAMETER EXTRACTION 2

CUSTOMIZING MODELS AND RESPONSE FUNCTIONS 3

STATISTICAL MODELING AND ANALYSIS 3

USER INTERFACE 4

COMPATIBILITY WITH OTHER CAD SOFTWARE 4

OSA SUPPORT 4

PLATFORMS AND AVAILABILITY 4

HarPE™ Versions 1.4/1.4+S Technical Summary

INTRODUCTION

HarPE™ has emerged from many years of theoretical research and incorporates benchmark results by Optimization Systems Associates Inc.

HarPE Version 1.4 is an integrated, advanced and powerful CAD system dedicated to device nonlinear simulation, characterization and optimization. HarPE Version 1.4+S additionally contains an optional statistical modeling and analysis module.

DEVICE SIMULATION

HarPE offers a comprehensive approach to device-oriented, seamlessly integrated large-signal, small-signal and DC simulations using a unified circuit description. When needed, the small-signal parameters are automatically linearized from the nonlinear model.

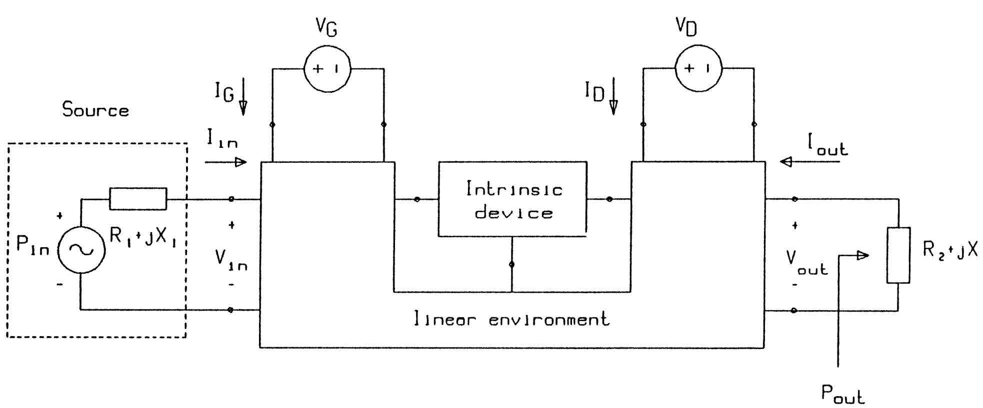
The overall schematic of a single-device circuit is shown below. P_{in} is the RF source described by its available power, R_1+jX_1 is the source impedance, P_{out} is the actual AC power delivered to the load R_2+jX_2, V_G represents the gate (base) DC bias voltage, and V_D the drain (collector) DC bias voltage. A one-port configuration is treated similarly.

At AC, HarPE calculates the input and output voltages and currents V_{in}, V_{out}, I_{in} and I_{out}, output power P_{out}, and small-signal S-parameters. At DC, HarPE calculates the currents at the bias ports, namely I_G and I_D.

Arbitrary combinations of bias, frequency and power sweeps are available. Parameter sweeps can also be simulated.

Built-in responses for high-quality graphical display and numerical output include:

- frequency spectra of powers, voltages and currents
- time-domain waveforms of voltages and currents
- spectrum responses vs. sweep of available input power or vs. sweep of fundamental excitation frequency
- DC IV curves
- small-signal parameters linearized from nonlinear models
- Smith chart and polar plots of S-parameters
- maximum available gain and stability factor
- sensitivity displays



Schematic representation of a single-device circuit

CIRCUIT DESCRIPTION

To represent the intrinsic part of the circuit, you can use one of the built-in nonlinear intrinsic models to achieve maximum computational efficiency. Currently, the nonlinear intrinsic device model library includes:

- the semiconductor diode model
- the Curtice and Ettenberg FET model
- the Materka and Kacprzak FET model
- the Raytheon (Statz *et al.*) FET model
- the physics based Khatibzadeh and Trew FET model
- the Gummel and Poon model of bipolar NPN and PNP transistors

Alternatively, you can create custom device models using HarPE's unique Expression Block.

To represent the linear environment you can use one of the predefined extrinsic super-components for extremely efficient linear simulations. Alternatively, you can define your own linear environment using built-in linear elements or supplying externally obtained measurement or simulation data.

HarPE accepts general complex, frequency dependent and user-defined terminations to provide for maximum flexibility for the simulation environment you need.

OPTIMIZATION

The power of HarPE lies in its implementation of state-of-the-art techniques. The gradient-based minimax, ℓ_1 and least-squares optimizers have a proven track record in circuit optimization.

Combining the optimizers with OSA's unique nonlinear FAST™ adjoint sensitivity analysis, HarPE provides solutions to your design and modeling problems with unparalleled speed, accuracy, consistency and robustness.

The functions, goals and measured data for optimization can be any combination of power spectra, waveforms, S-parameters and DC IV data. Stability factor and maximum available gain can also be optimized.

In addition to built-in responses, arbitrary user-defined functions can be optimized. Even the goals and window specifications can be user-defined expressions. This capability makes HarPE a flexible system for your modeling and design optimization needs.

PARAMETER EXTRACTION

Employing a unique multi-circuit concept, HarPE processes simultaneously multi-bias, multi-frequency, and multi-input-power-level measurements. This approach increases model identifiability and ensures a reliable and consistent solution suitable for large-signal, small-signal and DC simulation. In spite of the fact that numerical optimization of multiple nonlinear circuits is an extremely challenging task, HarPE stands out as the world's first software boasting this capability.

Parameter Extraction from Harmonic Data

HarPE is the world's first software system for parameter extraction from harmonic data. It offers a truly nonlinear device parameter extraction procedure which utilizes spectrum measurements, including DC bias information and power output at different harmonics measured under RF large-signal excitations.

The HarPE criterion for parameter extraction is that the computed and measured responses must simultaneously and optimally match at DC, fundamental frequency and higher harmonics. This approach directly exploits the nonlinear behavior of the device under test and ensures the most reliable solutions of large-signal model parameters for subsequent circuit simulation with circuit simulators.

HarPE experiments have shown that all device parameters can be uniquely identified under actual high-frequency large-signal operating conditions, demonstrating the importance of higher harmonics in large-signal parameter extraction.

Parameter Extraction from S-parameters and/or DC Data

HarPE can extract the nonlinear model parameters from small-signal S-parameter data or even just from DC data.

In fact, HarPE accepts device measurements in the form of any combination of power spectra, waveforms, small-signal S-parameters and/or DC data. The nonlinear device model automatically and smoothly links the model at all available bias points.

CUSTOMIZING MODELS AND RESPONSE FUNCTIONS

Using Fortran-like expressions, you can create your own device models or modify the built-in models to define process, geometrical, physical or abstract functional dependence of any model parameters. You can also define your own response functions, as well as goals and specifications for optimization. All these are achieved within HarPE's unique Expression Block. No subroutines need to be written, compiled or linked. This allows you to test, verify and even optimize in-house models effortlessly.

User-defined models and response functions enjoy the power of HarPE in simulation and optimization just like the built-in models and responses.

Any combinations of algebraic operations and standard mathematical functions are permitted in formulating an expression.

Conditional expressions are also supported. A set of predefined labels allows you to refer to the program's internal variables and built-in responses, such as the frequency, bias voltages, instantaneous intrinsic voltages, input power, output power spectrum components, DC current and S-parameters.

HarPE offers postprocessing of the simulation results for arbitrary user-defined response functions. You may use any combination of algebraic operations and standard mathematical functions to manipulate the simulated large- and small-signal responses in order to create your own response functions for graphical display, numerical output and optimization.

For example, you may define a custom response function such as

```
POWER_RATIO_DBM = POUT1 - PIN;
```

where PIN and POUT1 are predefined labels for the input power and the output power at fundamental frequency in dBm, respectively.

STATISTICAL MODELING AND ANALYSIS

Advanced features of statistical modeling and analysis are available as an optional add-on module in HarPE Version 1.4+S.

Any of the built-in or user defined models can be used for statistical modeling and analysis. HarPE is the first commercial software system to offer a physics based FET model *together with* statistical capabilities.

Multi-Device Modeling

Based on a large set of measurement data from multiple devices, a multi-device parameter extraction procedure provides a sample of device models. Histograms showing parameter spreads and scatter diagrams showing parameter correlations are displayed. The sample of device models can be further postprocessed to create a consolidated statistical model, immediately suitable for statistical Monte Carlo analysis.

Parameter correlations are included in the model. For parameters exhibiting substantially non-Gaussian distributions, the user can keep the histograms as discrete approximations of the marginal density functions.

Monte Carlo Analysis

HarPE Version 1.4+S features statistical Monte Carlo analysis. Uniform, normal, exponential and lognormal distributions with absolute or relative tolerances can be used. Histograms, run charts, and sweep diagrams are displayed. Yield can be computed from specifications on spectra, S-parameters, DC currents and arbitrary user-defined functions.

USER INTERFACE

The sophisticated capabilities of HarPE are presented through a polished and friendly user-interface. It is menu-driven, supports a mouse and features an integrated full screen editor, clear syntax, flexible data format, high-quality graphics, on-line help and on-line User's Manual. The following highlight some of the features.

- Logically designed menu-driven and window-driven commands with on-line help for every command.
- Mouse support facilitates quick and fluent operation. However, the program is fully operational from a keyboard without a mouse.
- Built-in dual-window, dual-file full screen editor with search and replace, cut and paste, buffered undo, macros, and more.
- File parser fully integrated into the screen editor. The location of any syntax error is automatically highlighted on screen by the editor.
- Clear and uncomplicated input file syntax with flexible and user-designable data format.
- On-line User's Manual. You can add your own comments to the Manual, or copy syntax definitions and examples in the Manual and paste them directly into your circuit file, or define and search for arbitrary keywords (dynamic indexing).
- The program can be operated in three different modes: run interactively, driven by a macro file, or run as a silent background process.
- Automatically generated log file maintains a complete record of the operations, including clock time, user commands and messages.

COMPATIBILITY WITH OTHER CAD SOFTWARE

HarPE can help you make the most effective use of other CAD products by supplying model parameters that truly belong to your own devices.

HarPE supports linear subcircuits described by "black box" S, Y, or Z data imported from external simulators.

In addition to its flexible data format, HarPE directly accepts measured data from Cascade Microtech's MicroCAT™ Test Executive system and data in the MDIF (Measurement Data Interchange Format) format.

OSA SUPPORT

As a user of HarPE, you can rely on the expedient and professional support from OSA's technical experts, including the authors of the program.

With your acquisition of HarPE you are automatically entitled to a standard 90 day warranty. An option of free software upgrades and maintenance for one year is also available.

PLATFORMS AND AVAILABILITY

HarPE Version 1.4 and HarPE Version 1.4+S are available on Apollo, Hewlett-Packard and Sun workstations. For further information please call or write.

Optimization Systems Associates Inc.
P.O. Box 8083
Dundas, Ontario
Canada L9H 5E7

Tel 416 628 8228 Fax 416 628 8225

HarPE™
Versions 1.5/1.5+S Technical Brief

INTRODUCTION

HarPE™ Version 1.5 is an advanced, powerful, integrated CAD system dedicated to complete nonlinear device characterization, and single device circuit simulations and design optimizations. HarPE™ Version 1.5+S additionally contains an optional statistical modeling and analysis module.

DEVICE SIMULATION

HarPE offers a comprehensive approach to device-oriented, seamlessly integrated frequency-domain large-signal and small-signal as well as DC simulations, using a unified circuit description. When needed, the small-signal parameters are automatically linearized from the nonlinear model.

The overall HarPE circuit topology consists of an intrinsic nonlinear device encompassed by a linear environment to which are connected the RF and bias sources and the load. Typically, voltage bias sources are considered. An option of base current bias source exists to accommodate measurement setups used for bipolar devices. HarPE allows for both 1-port and 2-port circuit configurations.

At AC, HarPE calculates the RF input and output voltages and currents, output power, and small-signal S parameters. At DC, HarPE calculates the DC responses (typically currents) at the bias ports. Arbitrary combinations of bias, frequency, power and arbitrary parameter sweeps are available.

Built-in responses include:

- power, voltage and current spectra
- voltage and current waveforms
- DC IV curves
- small-signal parameters linearized from nonlinear models
- Smith chart and polar plots of S parameters
- insertion loss, maximum available gain and stability factor

CIRCUIT DESCRIPTION

To represent the nonlinear part of the circuit, you can use one of the built-in intrinsic device models:

- the semiconductor diode model
- the Curtice and Ettenberg FET model
- the Materka and Kacprzak FET model
- the Raytheon (Statz *et al.*) FET model
- the Khatibzadeh and Trew FET model
- the improved Trew FET model for uniform doping
- the Gummel and Poon model of bipolar NPN and PNP transistors

Alternatively, you can create custom device models using HarPE's unique Expression Block.

To represent the linear environment you can use one of the predefined extrinsic super-components for extremely efficient linear simulations. Alternatively, you can define your own linear environment using built-in linear elements or supplying externally obtained measurement or simulation data.

Purely linear circuits are also accepted by HarPE. All types of controlled sources and a number of standard linear components can be used within a general user-defined topology. They are available to create any linear model you wish to build.

HarPE accepts general complex, frequency dependent and user-defined terminations to provide for maximum flexibility for the simulation environment you need.

OPTIMIZATION

The power of HarPE lies in its implementation of state-of-the-art techniques. The gradient-based minimax, ℓ_1 and least-squares optimizers have a proven track record in circuit optimization.

Combining the optimizers with OSA's unique nonlinear FAST™ adjoint sensitivity analysis, HarPE provides solutions to your design and modeling problems with unparalleled speed, accuracy, consistency and robustness.

A distinct feature of sensitivity displays is available to help you to select the most crucial parameters for optimization.

In addition to built-in responses, arbitrary user-defined functions can be optimized. The goals and window specifications can be defined directly or through expressions. These capabilities make HarPE very flexible for your modeling and design optimization needs.

PARAMETER EXTRACTION

HarPE was introduced as the world's first software system for parameter extraction from harmonic data. It offers a truly nonlinear device parameter extraction procedure which utilizes spectrum measurements, including DC bias information and output power at different harmonics measured under RF large-signal excitations.

If large-signal measurements are not available HarPE can extract the nonlinear model parameters from small-signal S-parameter data taken over a number of bias conditions, or even just from DC data (which alone is not sufficient to determine some parameters such as nonlinear capacitors). In general, HarPE accepts and fully utilizes all information from any combination of the three types of measurement data.

CUSTOMIZING MODELS AND RESPONSE FUNCTIONS

Using typical mathematical expressions, you can create your own device models or modify the built-in models directly in the circuit file. Preprogrammed user-defined models include both the symmetric and asymmetric Curtice models, the Materka model, the Plessey model and TriQuint's Own Model (TOM).

You can also define your own response functions, as well as goals and specifications for optimization. User-defined models and response functions enjoy the power of HarPE in simulation and optimization just like the built-in models and responses.

Combinations of algebraic operations and standard mathematical functions are permitted in formulating an expression. Conditional expressions (nested IF ELSE structures) are also supported.

A set of predefined labels allows you to refer to the program's internal variables and built-in responses, such as the frequency, bias voltages, intrinsic voltages, input power, output power spectrum components, DC current and S-parameters.

USER INTERFACE

The sophisticated capabilities of HarPE are presented through a polished and friendly user interface. It is menu-driven, supports a mouse and features an integrated full screen editor, clear syntax, flexible and user-designable data format, high-quality graphics, on-line help and on-line User's Manual. Text substitution (macro definitions) allows you to introduce your own keywords, circuit description, etc., according to your preference, language, or requirements of other programs that such a description may be imported to or exported from. You can also select your own colour map.

COMPATIBILITY WITH OTHER CAD SOFTWARE

HarPE's parameter extraction capabilities can help you make the most effective use of other CAD software by supplying model parameters that truly belong to the devices for which you have access to measurement data. HarPE's expression processor allows you to modify the built-in models or create user-defined models to easily adapt to any new developments or modifications introduced by other vendors. This ensures that HarPE models are always compatible with other software you use.

HarPE supports linear subcircuits described by "black box" S, Y, or Z data imported from external simulators.

In addition to its flexible data format, HarPE directly accepts measured data from Cascade Microtech's MicroCAT™ Test Executive system and data in the MDIF (Measurement Data Interchange Format) format.

STATISTICAL MODELING AND ANALYSIS

An optional add-on module for statistical modeling and analysis is available in HarPE Version 1.5+S.

Any of the built-in or user defined models can be used for statistical modeling. Based on a large set of measurement data from multiple devices, a multi-device parameter extraction procedure provides a sample of device models. This can be further postprocessed automatically in order to create a consolidated statistical model, immediately ready for statistical Monte Carlo analysis.

OSA SUPPORT

With your acquisition of HarPE you are entitled to a standard 90 day warranty.

An option of free software program upgrades and maintenance for one year is also available. To continue to receive these benefits after the first year you may purchase an extended software support option.

PLATFORMS AND AVAILABILITY

HarPE Versions 1.5 and 1.5+S are available on Apollo, Hewlett-Packard and Sun workstations. For further information please call or write.

Optimization Systems Associates Inc.
P.O. Box 8083, Dundas, Ontario
Canada L9H 5E7

Tel 416 628 8228 Fax 416 628 8225

HarPE™
Version 1.7 Technical Brief

INTRODUCTION

HarPE™ Version 1.7 is a powerful CAD software system dedicated to complete nonlinear device characterization including advanced statistical modeling and parameter extraction. HarPE provides you with simulation and design optimization capabilities for single device circuits.

DEVICE SIMULATION

The overall HarPE circuit topology consists of an intrinsic nonlinear device encompassed by a linear environment to which are connected the RF and bias sources and the load. Typically, voltage bias sources are considered. The option of base current bias source exists to accommodate measurement setups used for bipolar devices. HarPE allows for both one-port and two-port circuit configurations.

At AC, HarPE calculates the RF input and output voltages and currents, output power, and small-signal S parameters. At DC, HarPE calculates the DC responses (typically currents) at the bias ports. The simulation ranges can be defined by bias, frequency, power and arbitrary parameter sweeps.

Built-in responses include:

- power, voltage and current spectra
- voltage and current waveforms
- DC IV curves
- small-signal parameters linearized from nonlinear models
- Smith chart and polar plots of S parameters
- insertion loss, maximum available gain and stability factor

HarPE offers a comprehensive approach to seamlessly integrated large- and small-signal frequency-domain simulations of nonlinear devices. Using a unified circuit description the small-signal parameters of nonlinear devices, when needed, are automatically derived from nonlinear models.

CIRCUIT DESCRIPTION

To represent the nonlinear part of the circuit, you can use one of the built-in intrinsic device models:

- the semiconductor diode model
- the Curtice and Ettenberg FET model
- the Materka and Kacprzak FET model
- the Raytheon (Statz *et al.*) FET model
- the Khatibzadeh and Trew FET model
- the modified Trew FET model for uniform doping
- the KTL (Trew/Ladbrooke) FET model
- the Gummel and Poon model of bipolar NPN and PNP transistors

Alternatively, you can create custom device models using HarPE's unique Expression Block.

To represent the linear environment you can use one of the predefined extrinsic super-components for extremely efficient linear simulations. Alternatively, you can define your own linear environment using built-in linear elements or supplying external measurement or simulation data. The library of linear elements includes a comprehensive set of microstrip structures, including typical discontinuities, as well as a number of lumped elements such as resistors, capacitors, inductors and controlled sources.

Purely linear circuits are also accepted by HarPE. All linear components can be used within a general user-defined topology. This is particularly useful in defining and extracting parameters of small-signal models.

HarPE accepts general complex, frequency-dependent and user-defined terminations to provide for maximum flexibility for the simulation environment you need.

OPTIMIZATION

The power of HarPE lies in its implementation of state-of-the-art techniques. The gradient-based minimax, ℓ_1 and least-squares optimizers have proven track records in circuit optimization.

Combining the optimizers with OSA's unique nonlinear FAST™ adjoint sensitivity analysis, HarPE provides solutions to your design and modeling problems with unparalleled speed, accuracy, consistency and robustness.

An extremely useful feature of sensitivity analysis can assist you, via easy-to-understand graphical display, in selecting the most crucial parameters for optimization.

In addition to built-in responses, arbitrary user-defined functions can be optimized. The optimization goals as well as window specifications can be defined directly or using expressions. This makes HarPE very flexible for all your modeling needs.

PARAMETER EXTRACTION

HarPE was introduced as the world's first software system for parameter extraction from harmonic data. It offers a truly nonlinear device parameter extraction procedure which utilizes spectrum measurements, including DC bias information and output power at different harmonics measured under RF large-signal excitations.

If large-signal measurements are not available HarPE can extract the nonlinear model parameters from small-signal S-parameter data taken over a number of bias conditions, or even just from DC data (which alone is not sufficient to determine some parameters such as nonlinear capacitors). In general, HarPE accepts and fully utilizes all information from any combination of the three types of measurement data.

CUSTOMIZING MODELS AND RESPONSE FUNCTIONS

Using typical mathematical expressions, you can create your own device models or modify the built-in models directly in the circuit file. Preprogrammed user-defined models include both the symmetric and asymmetric Curtice models, the Materka model, the Plessey model and TriQuint's Own Model (TOM), all suitable for both DC and AC simulations.

You can also define your own response functions, as well as goals and specifications for optimization. User-defined models and response functions enjoy the power of HarPE in simulation and optimization just like the built-in models and responses.

Combinations of algebraic operations and standard mathematical functions are available in formulating an expression. Conditional expressions (nested IF ELSE structures) are also supported.

A set of predefined labels allows you to refer to the program's internal variables and built-in responses, such as the frequency, bias voltages, intrinsic voltages, input power, output power spectrum components, DC current and S parameters.

USER INTERFACE

The sophisticated capabilities of HarPE are presented through a polished and friendly user interface. It is menu-driven, supports mouse operation and features an integrated full screen editor, clear syntax, flexible and user-designable data format, high-quality graphics, on-line help and on-line User's Manual.

Text macro definitions allow you to introduce your own keywords, circuit description, etc., according to your preference, language, or requirements of other programs that such a description may be imported to or exported from. You can also select your own colour map.

STATISTICAL MODELING AND ANALYSIS

Statistical modeling, as a prerequisite for meaningful statistical analysis and yield optimization, is one of the most significant features of HarPE.

Any of the built-in or user defined models can be used for statistical modeling. Based on a large set of measurement data from multiple devices, a multi-device parameter extraction procedure provides a sample of device models. This can be further postprocessed automatically in order to create a consolidated statistical model, immediately ready for statistical Monte Carlo analysis.

COMPATIBILITY WITH OTHER CAD SOFTWARE

HarPE's parameter extraction capabilities can help you use other CAD software most effectively by supplying model parameters that truly belong to the devices for which you have access to measurement data. All built-in library models in HarPE are fully compatible with those in OSA90/hope™ - a general CAD software system from OSA.

HarPE's expression processor allows you to modify the built-in models or create user-defined models to easily adapt to any new developments or modifications introduced by other vendors. This ensures that HarPE models are always compatible with other software you use.

HarPE supports linear subcircuits described by "black box" S, Y, or Z data imported from external simulators.

In addition to its flexible data format, HarPE directly accepts measured data from Cascade Microtech's MicroCAT™ Test Executive system and data in the MDIF (Measurement Data Interchange Format) format.

WARRANTY

With your acquisition of HarPE you are entitled to a 90 day limited warranty. A software support option which includes software upgrades is also available.

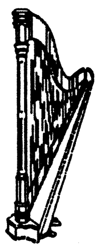
As a user of HarPE, you can rely on professional and timely support from our technical experts, including authors of the program.

PLATFORMS AND AVAILABILITY

HarPE runs under X windows on Hewlett-Packard and Sun workstations. For further information contact

Optimization Systems Associates Inc.
P.O. Box 8083, Dundas, Ontario
Canada L9H 5E7

Before Oct. 4/93 Tel 416 628 8228 Fax 416 628 8225
After Oct. 4/93 Tel 905 628 8228 Fax 905 628 8225



HarPE™
Version 1.8 Technical Brief

INTRODUCTION

HarPE™ is a powerful CAD software system dedicated to complete nonlinear device characterization including advanced statistical modeling and parameter extraction. HarPE provides you with simulation and design optimization capabilities for single nonlinear device circuits.

DEVICE MODELS

The circuit topology for HarPE consists of an intrinsic nonlinear device encompassed by an extrinsic linear environment representing the parasitic, packaging and bias circuits. The overall circuit configuration can be either a one-port or a two-port.

The built-in intrinsic nonlinear device models include:

- the semiconductor diode model
- the Curtice and Ettenberg FET model
- the Materka and Kacprzak FET model
- the Raytheon (Statz *et al.*) FET model
- the Khatibzadeh and Trew FET model
- the modified Trew FET model for uniform doping
- the KTL (Trew/Ladbrooke) FET model
- the Gummel and Poon model of bipolar NPN and PNP transistors
- HEMT models (HOBD, DP, Curtice) provided by Golio, Motorola
- custom device models created using HarPE's unique Expression Block

The linear environment can be represented by built-in extrinsic super-components for the fastest simulation speed. It can also be constructed from linear library elements or supplied as *S*, *Y* or *Z* data.

The linear library elements includes the typical lumped elements, a comprehensive set of microstrip structures and controlled sources.

CIRCUIT SIMULATION

At AC, HarPE calculates the RF input and output voltages and currents, output power, and small-signal *S* parameters. At DC, HarPE calculates the DC currents and voltages at the bias ports. Frequency, bias, power and arbitrary parameter sweeps can be defined for simulation.

Built-in responses include:

- power, voltage and current spectra
- voltage and current waveforms
- DC IV curves
- small-signal parameters linearized from nonlinear models
- Smith chart and polar plots of S parameters
- insertion loss, maximum available gain and stability factor

HarPE offers a comprehensive approach to seamlessly integrated large- and small-signal frequency-domain simulations of nonlinear devices. Using a unified circuit description the small-signal parameters of nonlinear devices, when needed, are automatically derived from nonlinear models.

OPTIMIZATION

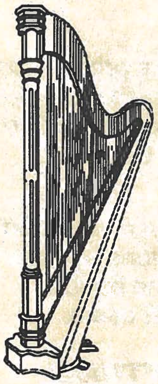
The power of HarPE lies in its implementation of state-of-the-art techniques. The gradient-based minimax, ℓ_1 and least-squares optimizers have proven track records in circuit optimization. A novel Huber optimizer enhances Version 1.8.

Combining the optimizers with OSA's unique nonlinear FAST™ adjoint sensitivity analysis, HarPE provides solutions to your design and modeling problems with unparalleled speed, accuracy, consistency and robustness.

PARAMETER EXTRACTION

HarPE was introduced as the world's first software system for parameter extraction from harmonic data. It offers a truly nonlinear device parameter extraction procedure which utilizes spectrum measurements, including DC bias information and output power at different harmonics measured under RF large-signal excitations.

If large-signal measurements are not available HarPE can extract the nonlinear model parameters from small-signal S-parameter data taken over a number of bias conditions, or even just from DC data (which alone is not sufficient to determine some parameters such as nonlinear capacitors). In general, HarPE accepts and fully utilizes all information from any combination of the three types of measurement data.



HarPE™ Version 2.0 Technical Brief

INTRODUCTION

HarPE™ is a powerful CAD software system dedicated to complete nonlinear device characterization including advanced statistical modeling and parameter extraction. **HarPE** provides you with simulation and design optimization capabilities for single nonlinear device circuits.

DEVICE MODELS

The circuit topology for **HarPE** consists of an intrinsic nonlinear device encompassed by an extrinsic linear environment representing the parasitic, packaging and bias circuits. The overall circuit configuration can be either a one-port or a two-port.

The built-in intrinsic nonlinear device models include:

- the semiconductor diode model
- the Curtice and Ettenberg FET model
- the Materka and Kacprzak FET model
- the Raytheon (Statz *et al.*) FET model
- the Khatibzadeh and Trew FET model
- the modified Trew FET model for uniform doping
- the KTL (Trew/Ladbrooke) FET model
- the Gummel and Poon model of bipolar NPN and PNP transistors
- HEMT models (HOBD, DP, Curtice) provided by Golio, Motorola
- the HBT model described by Wang *et al.*, CNET
- custom device models created using **HarPE**'s unique Expression Block

The linear environment can be represented by built-in extrinsic super-components for the fastest simulation speed. It can also be constructed from linear library elements or supplied as *S*, *Y* or *Z* data.

The linear library elements includes the typical lumped elements, a comprehensive set of microstrip structures and controlled sources.

CIRCUIT SIMULATION

At AC, **HarPE** calculates the RF input and output voltages and currents, output power, and small-signal *S* parameters. At DC, **HarPE** calculates the DC currents and voltages at the bias ports. Frequency, bias, power and arbitrary parameter sweeps can be defined for simulation.

Built-in responses include:

- power, voltage and current spectra
- voltage and current waveforms
- DC IV curves
- small-signal parameters linearized from nonlinear models
- Smith chart and polar plots of S parameters
- insertion loss, maximum available gain and stability factor

HarPE offers a comprehensive approach to seamlessly integrated large- and small-signal frequency-domain simulations of nonlinear devices. Using a unified circuit description the small-signal parameters of nonlinear devices, when needed, are automatically derived from nonlinear models.

OPTIMIZATION

The power of **HarPE** lies in its implementation of state-of-the-art techniques. The gradient-based minimax, ℓ_1 and least-squares optimizers have proven track records in circuit optimization. A novel Huber optimizer offers enhanced flexibility in modeling.

Combining the optimizers with OSA's unique nonlinear FAST™ adjoint sensitivity analysis, **HarPE** meets your design and modeling needs with unparalleled speed, accuracy, consistency and robustness.

Optimization specifications can be entered directly on the graphical display of the response of interest.

PARAMETER EXTRACTION

HarPE was introduced as the world's first software system for parameter extraction from harmonic data. It offers a truly nonlinear device parameter extraction procedure which utilizes spectrum measurements, including DC bias information and output power at different harmonics measured under RF large-signal excitations.

If large-signal measurements are not available **HarPE** can extract the nonlinear model parameters from small-signal S-parameter data taken over a number of bias conditions, or even just from DC data (which alone is not sufficient to determine some parameters such as nonlinear capacitors). In general, **HarPE** accepts and fully utilizes all information from any combination of the three types of measurement data.

PREPROCESSING "COLD" MEASUREMENTS

Version 2.0 offers the option of extracting the extrinsic parameters analytically from cold measurements (unbiased and pinched-off measurements). The extracted extrinsic parameters are then kept fixed when the intrinsic device parameters are extracted from hot measurements.

CUSTOMIZING MODELS AND RESPONSES

You can create new device models or customize existing ones using algebraic expressions and mathematical functions. A set of preprogrammed models can be used as a template. Variables which may be necessary for user-defined models, such as the frequency, bias voltages and intrinsic voltages, are accessible through built-in labels. Conditional expressions (IF ELSE structures) are supported.

Expressions can also be used to create user-defined response functions, which can then be displayed and optimized just like the built-in responses.

The flexibility of customizing device models and responses has been instrumental for many **HarPE** users in developing new and/or proprietary FET, bipolar, HEMT and HBT models, including modeling device thermal characteristics.

STATISTICAL MODELING AND ANALYSIS

Statistical modeling is a prerequisite for realistic yield analysis and optimization.

HarPE implements a multi-device parameter extraction procedure for statistical modeling. From a set of multi-device measurements, a corresponding set of device models is extracted. This sample of device models is automatically postprocessed to create a consolidated statistical model, ready for Monte Carlo simulation.

Another novel approach offered by **HarPE** is statistical modeling by cumulative probability distribution (CPD) fitting. Model parameter statistics are directly extracted by matching the model and measured response distributions. Response histogram matching (based on Chi-square test) is also available.

The result of Monte Carlo analysis can be displayed as sweep plots, histograms, run charts, yield, scattering diagrams and cumulative probability distribution plots.

HarPE AS A DATAPIPE™ CHILD OF OSA90/hope™

HarPE Version 2.0 can be invoked from **OSA90/hope** as a Datapipe child. This can dynamically integrate the comprehensive general-purpose circuit-level features of **OSA90/hope** with the efficient and dedicated device modeling capabilities of **HarPE**. It also opens the door to combining active device simulation and electromagnetic field simulation using **Empipe™** in circuit designs for first-pass success.

USER INTERFACE

The sophisticated capabilities of **HarPE** are presented through a polished and friendly user interface under X-Windows. It is menu-driven and features an integrated full screen editor, on-line help and on-line User's Manual.

Text macro definitions allow you to introduce your own keywords, circuit description, etc. The color map, key map and "hot keys" are user-definable. The flexible data format is especially favored by **HarPE** users.

COMPATIBILITY WITH OTHER CAD SOFTWARE

HarPE's parameter extraction capabilities can help you use other CAD software most effectively by supplying model parameters that truly characterize the devices in your design. The library models in **HarPE** are fully compatible with those in **OSA90/hope** - our general-purpose circuit CAD software system.

HarPE's expression processor allows you to modify the built-in models or create user-defined models to easily adapt to new developments or modifications introduced by other vendors. This ensures that **HarPE** models are compatible with other software you use.

HarPE supports linear subcircuits described by "black box" *S, Y* or *Z* data imported from external simulators.

In addition to its flexible data format, **HarPE** is able to accept directly *S*-parameter data files in the Touchstone® format or the MDIF (Measurement Data Interchange Format) format, as well as on-wafer measurement data produced by Cascade Microtech's MicroCAT™ Test Executive system.

WARRANTY AND SUPPORT

With your acquisition of **HarPE** you are entitled to a 90 day limited warranty. A software support option which includes software upgrades is also available.

As a user of **HarPE**, you can rely on professional and timely support from our technical experts, including authors of the program.

PLATFORMS AND AVAILABILITY

HarPE runs under X-Windows on Hewlett-Packard, Sun and DEC workstations. For further information contact

Optimization Systems Associates Inc.
P.O. Box 8083, Dundas, Ontario
Canada L9H 5E7

Tel 905 628 8228
Fax 905 628 8225