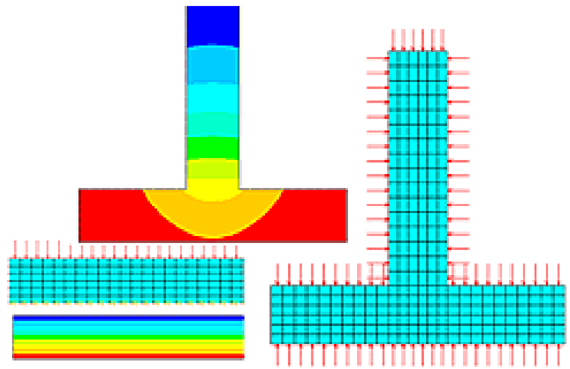
The Focus
PADT
A Publication for ANSYS Users

Contents

Feature Articles

- ANSYS Thermal: Getting the Heat Flow from Convection BCs
- Benchmarking ANSYS 6.1 on HP Systems
- CADfix



CADfix®

On the Web

- New in ANSYS 6.1
- PADT Seminar: Multiphysics Analysis
- PADT Seminar: Verifying FE Solutions / "Model Checker"
- Geometry Healer Replaces CADfix for ANSYS

hp invent

Resources

- PADT Support: How can we help?
- Upcoming Training at PADT
- About *The Focus*
  - *The Focus* Library
  - Contributor Information
  - Subscribe / Unsubscribe
  - Legal Disclaimer
- Next in *The Focus*

ANSYS Support Distributor

The Focus PADT
A Publication for ANSYS Users

ANSYS Thermal   Getting the Heat Flow from Convection BCs

by J. Luis Rosales, Thermal and Fluid Analyst

The purpose of this article is to show users how to extract the heat flow entering or leaving a model on the boundaries of a domain where convection has been applied. This information can be very important because it may be necessary to know if the model is conserving energy. Currently, immediate heat flow information can be obtained from boundaries where a temperature DOF has been specified using the reaction solution but this is not available for convection boundaries. This article will present a simple macro that will quickly give heat flow information for convection boundaries. This macro follows the same procedure used in the element table to extract the heat flow, but it is much easier especially for users who are not familiar with the element table. Two very simple example problems will be used to illustrate the use of the macro. The first example will have both a temperature and a convection boundary condition and the second will have only convection boundaries.

Example Problem 1: Temperature and Convection on a Block

A schematic of the meshed problem domain is shown in Figure 1 below. The arrows along the upper surface depict the convection boundary condition, and the small arrows along the lower boundary surface depict the temperature boundary condition. A temperature of 100°C was imposed along the bottom while a convection film coefficient of 10 W/m-K and a bulk temperature of 20°C were used to simulate natural convection along the top boundary.

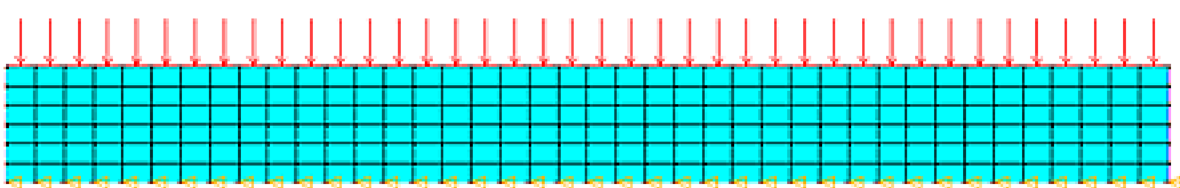


Figure 1. Schematic of the first example problem and the applied boundary conditions.

The Focus
A Publication for ANSYS Users

The thermal conductivity of aluminum was used for the material to perform a steady-state analysis. The resulting temperature contours are shown in Figure 2 below.

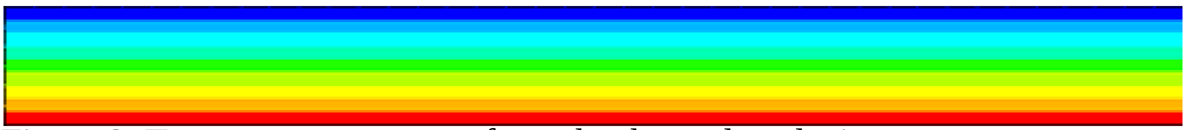


Figure 2. Temperature contours from the thermal analysis.

Since no boundary conditions were applied to the side surfaces, ANSYS/Thermal assumes they are adiabatic (no heat flow across boundary) by default. The temperature distribution varies very little since aluminum is a good conductor. The first step in examining the energy balance is to check the heat flow across the constant temperature boundary condition using the reaction solution. The reaction solution can be found under the following menu: General Postproc / List Results / Reaction Solu. Clicking OK in the small window with the Heat Flow selected will give a heat flow value of 1195.5 W. It is important to note that the reaction solution is only performed on nodes that are constrained by a temperature DOF. Therefore, the reaction solution cannot be used when there are no temperature constraints. Also, if more than one boundary is constrained by a temperature, the user can isolate the nodes on each boundary to determine the heat flow across that boundary. The next step is to determine the heat flow across the convection boundary surfaces. The heat flow can be extracted from the convection surface using the very simple macro shown below in Figure 3.

```
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!                                                         !
! MACRO THAT USES THE ELEMENT TABLE TO GET THE HEAT FLOW  !
! ACROSS A BOUNDARY WHERE CONVECTION HAS BEEN APPLIED.    !
!                                                         !
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!
 ETABLE,,NMISC, 5    ! BOTTOM SURFACE
 ETABLE,,NMISC,11    ! RIGHT SURFACE
 ETABLE,,NMISC,17    ! TOP SURFACE
 ETABLE,,NMISC,23    ! LEFT SURFACE
!
 SSUM
```

Figure 3. Macro for the extraction of convection heat flow.

The Focus
A Publication for ANSYS Users

Note that the NMISC value of 5 is used to obtain the convection heat flow from the bottom surface for a Plane 55 element. Similarly, a value of 11, 17, and 23 is used to extract the heat flow value for the right, top and left surfaces, respectively. When the macro from Figure 3 is read into ANSYS, the following lines of text will be written to the black output window.

Table Label	Total
NMIS5	0.00000
NMIS11	0.00000
NMIS17	1195.53
NMIS23	0.00000

Notice that the only label that contains a heat flow value is NMIS17 because a convection boundary condition was only applied along the upper surface. The value given by the macro using element table commands is 1195.53 W, which is the value we obtained from the reaction solution on the temperature nodes. Since energy enters through the temperature nodes and exits by convection, the model clearly is conserving energy.

Example Problem 2: Convection on a Fin

A schematic of the meshed fin problem is shown in Figure 4. The model corresponds to a small segment of the upper wall of a long channel with fins. In this model, a convection boundary condition is applied along the lower surface of the channel wall to simulate the flow of hot gas. The gas temperature is set to 150°C with a convection coefficient of 50 W/m-K. Longitudinal conduction is assumed negligible; therefore, the two side surfaces are left as adiabatic. The two upper wall surfaces and the three fin areas have a convection coefficient of 45 W/m-K and the external bulk fluid temperature is set to 20°C.

The Focus
PADT
A Publication for ANSYS Users

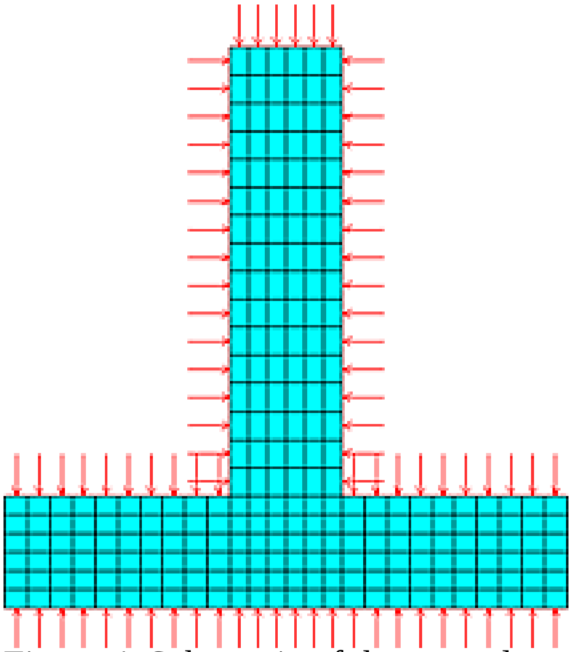


Figure 4. Schematic of the second example problem.

The thermal conductivity for aluminum is again used for the wall material property. The energy entering through the lower surface will conduct through the wall and the fin and leave the model through convection into the cool air. The temperature contours obtained from the thermal analysis is shown in Figure 5. The presence of the fin on the wall helps reduce the temperature at the fin base because of the added fin area exposed to the cool air. Unfortunately, this model has no temperature boundaries and thus, the reaction solution cannot be used to quickly extract the heat flow at those boundaries. Again, the simple macro shown in Figure 3 can be used to determine the heat flow through the boundaries of the model.

The Focus
PADT
A Publication for ANSYS Users

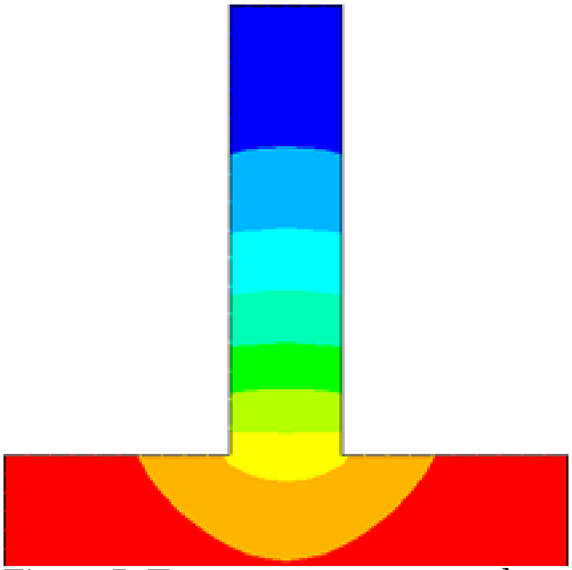


Figure 5. Temperature contours along the fin geometry.

The heat flow values are easily obtained by reading the macro into ANSYS. Again, the following results will be written to the ANSYS Output Window.

Table Label	Total
NMIS5	-447.182
NMIS11	134.806
NMIS17	177.570
NMIS23	134.806

Revisiting the manual for a Plane 55 element shows that `NMIS5` corresponds to the lower surface, `NMIS11` to the left surface, `NMIS17` to the top surface, and `NMIS23` to the right surface region of corresponding element. In this analysis a total of 447.182 W are entering the model through the lower surface (from the hot gas). A value of 134.806 W is leaving through each of the left and right surfaces of the model and a value of 177.570 W are leaving through the top surfaces of the model. Therefore, we can see that energy is once again conserved. The value of the heat flow leaving from the left and right surfaces of the fin is given by `NMIS11` and `NMIS23`, however, the value given by `NMIS17` contains the heat flow from the top wall surfaces and the top fin surface. To determine the heat flow from the top fin surface, the elements at that location must be isolated.

The heat flow from the surfaces of these example problems was easily determined because the models are mapped meshed so the elements are arranged in an orderly fashion. When a model is free meshed with triangular elements, the orientation of the elements is no longer easily determined. However, the resulting heat flow can

The Focus
PADT
A Publication for ANSYS Users

still be extracted with some care. The previous example can be free meshed as shown in Figure 6 below.

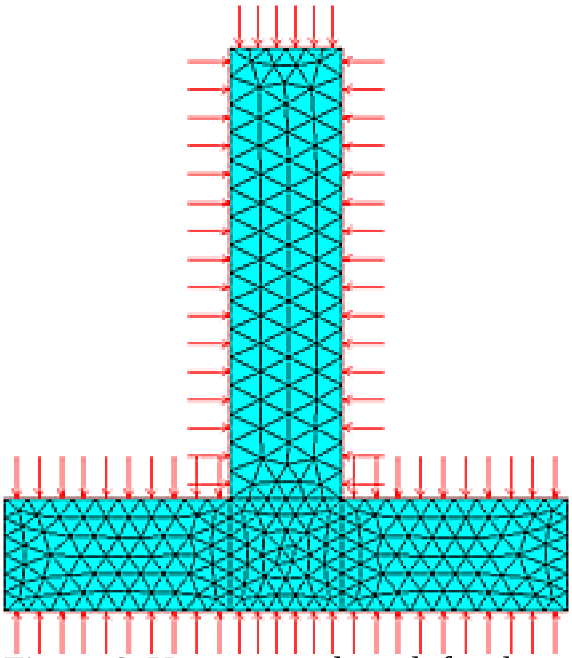


Figure 6. Unstructured mesh for the second example problem.

The same boundary conditions are applied and the resulting temperature contours are exactly as shown in Figure 5. When the macro is read into ANSYS the heat flow results are balanced but they are not the same as before. In order to determine the heat flow through the surfaces of the model, the elements corresponding to the surface lines must be isolated. By isolating the hot gas boundary elements and the cool gas boundary elements individually, the macro gives the following results:

Table Label	Total (*hot gas*)	Table Label	Total (*cool air*)
NMIS5	-396.495	NMIS5	387.820
NMIS11	-50.7105	NMIS11	59.3856
NMIS17	0.00000	NMIS17	0.00000
NMIS23	0.00000	NMIS23	0.00000

They both add up to 447.2056 W, which compares very well with the value of 447.182 W found for the structured model. Therefore, the macro can be used for any type of mesh.

The Focus PADT
A Publication for ANSYS Users

Conclusion

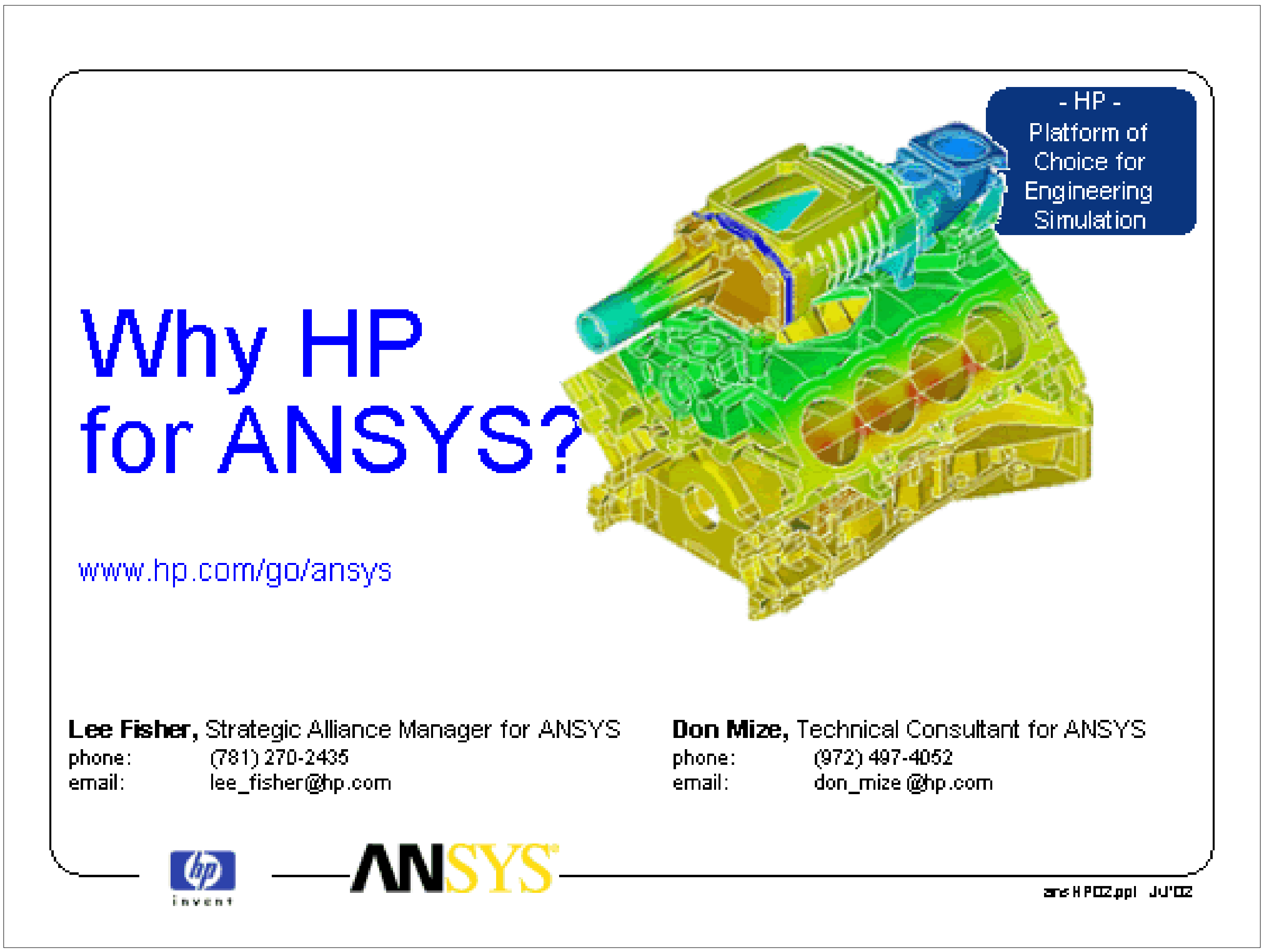
The two example problems illustrate the ease with which convection surface heat flows can be determined. If a temperature DOF is specified on a boundary, the reaction solution can quickly be used to determine the heat flow through those nodes. Although these macros were created for a 2D Plane 55 thermal elements, a user can easily build a similar macro for 3D Solid 70 thermal elements.

The Focus
A Publication for ANSYS Users

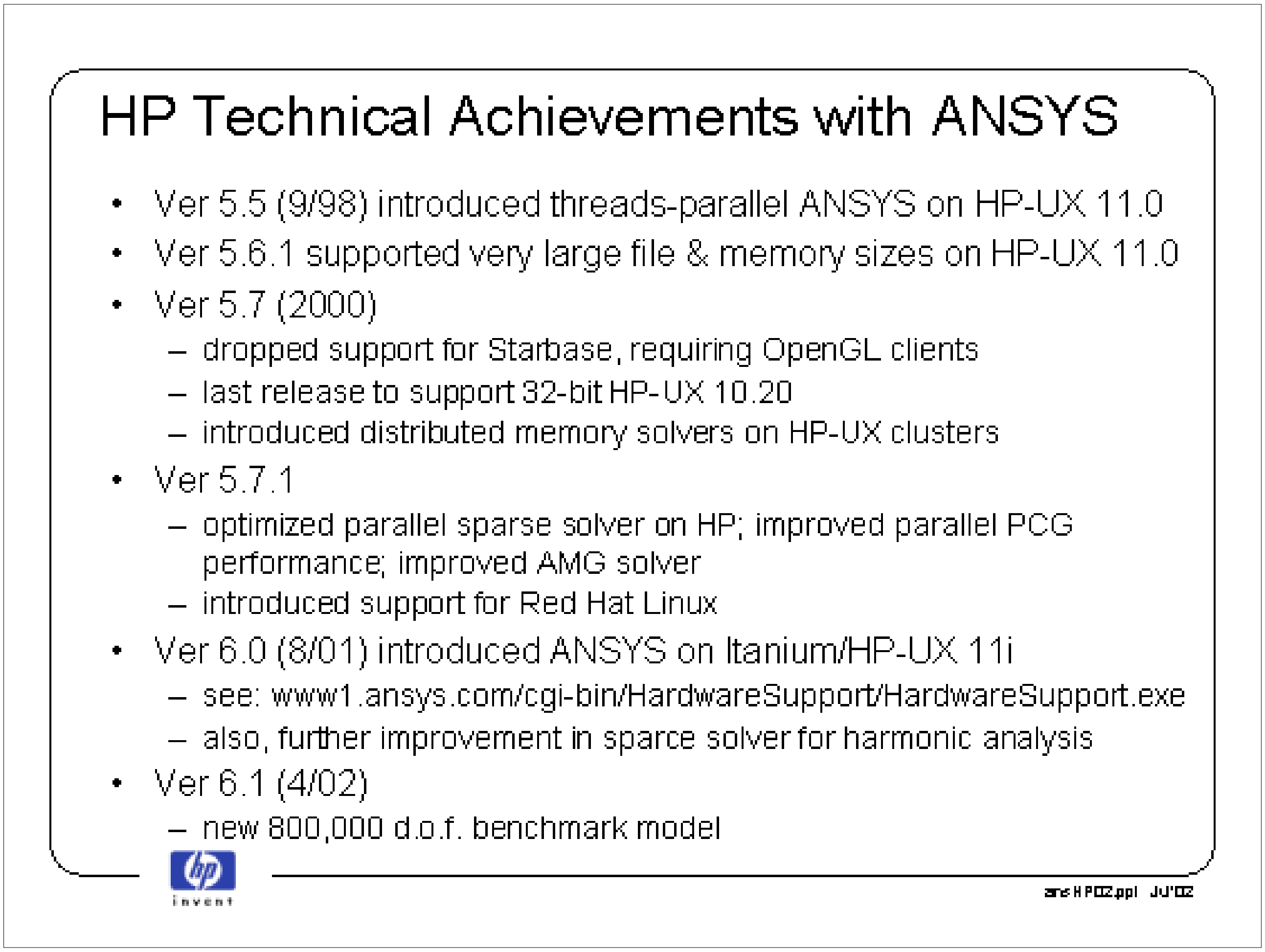
Benchmarking ANSYS 6.1 on HP Systems

by Lee Fisher, Hewlett-Packard

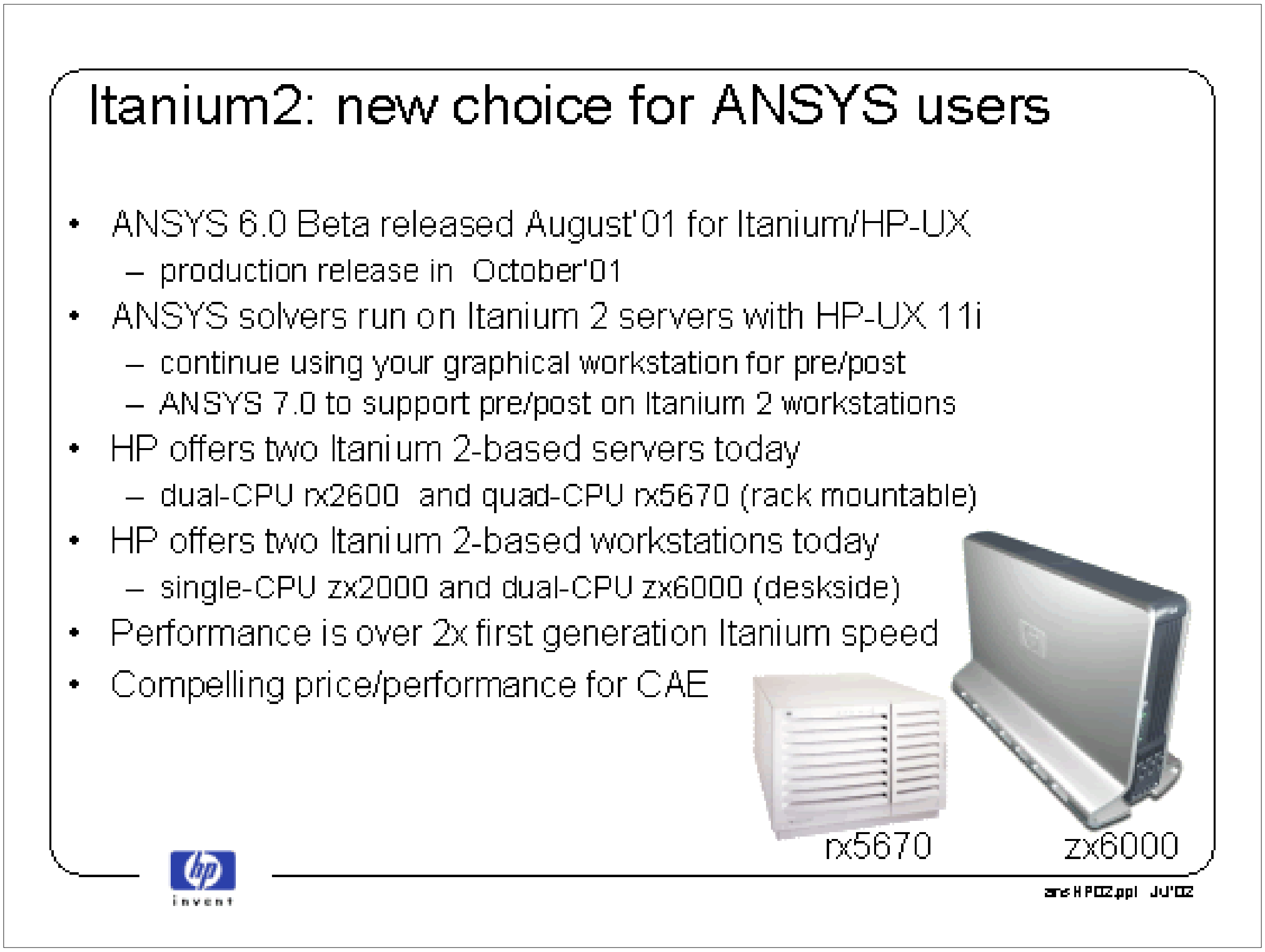
Note: The following Microsoft PowerPoint presentation may also be accessed directly here (531 KB).



The Focus
A Publication for ANSYS Users

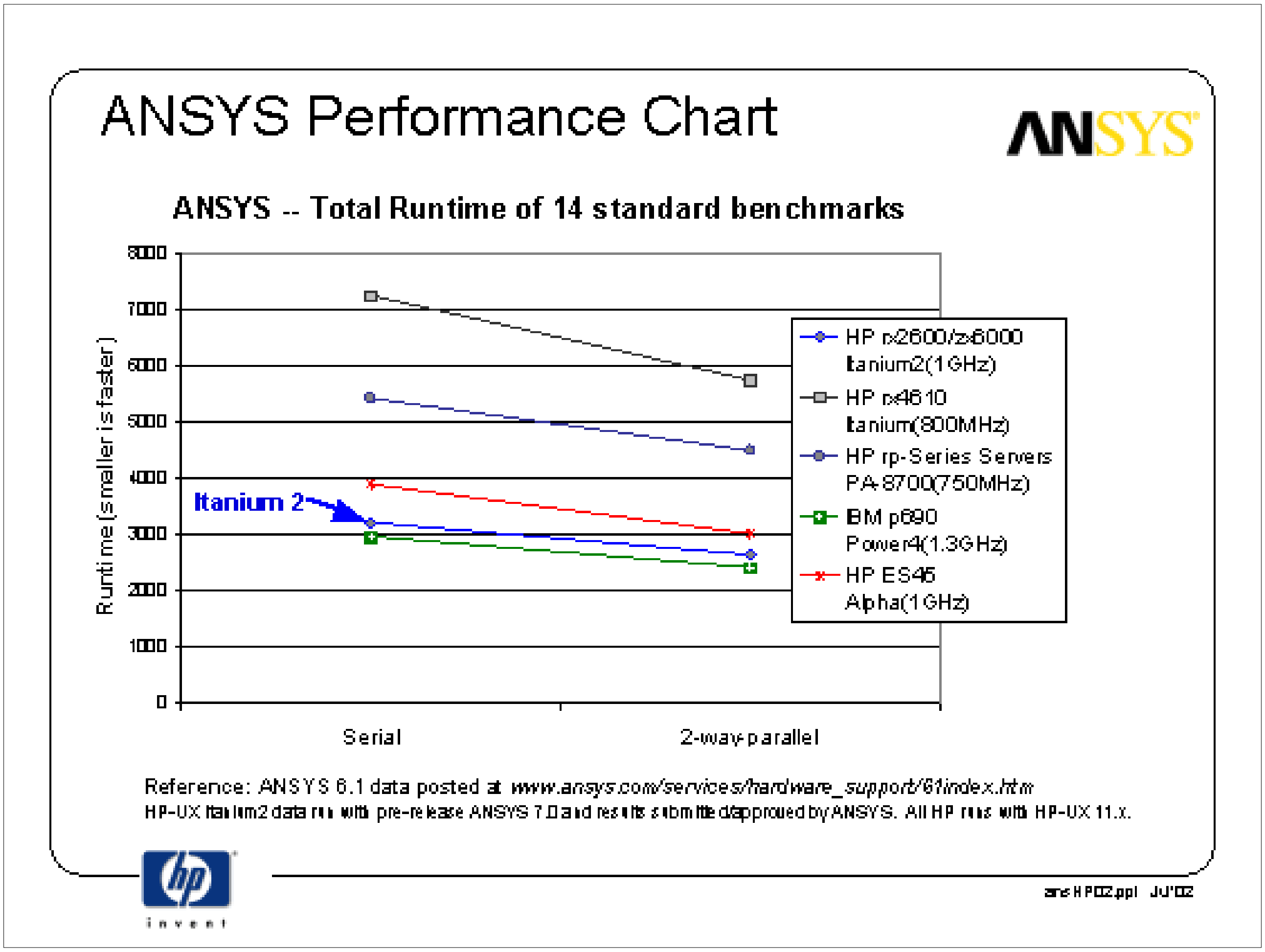


The Focus PADT
A Publication for ANSYS Users

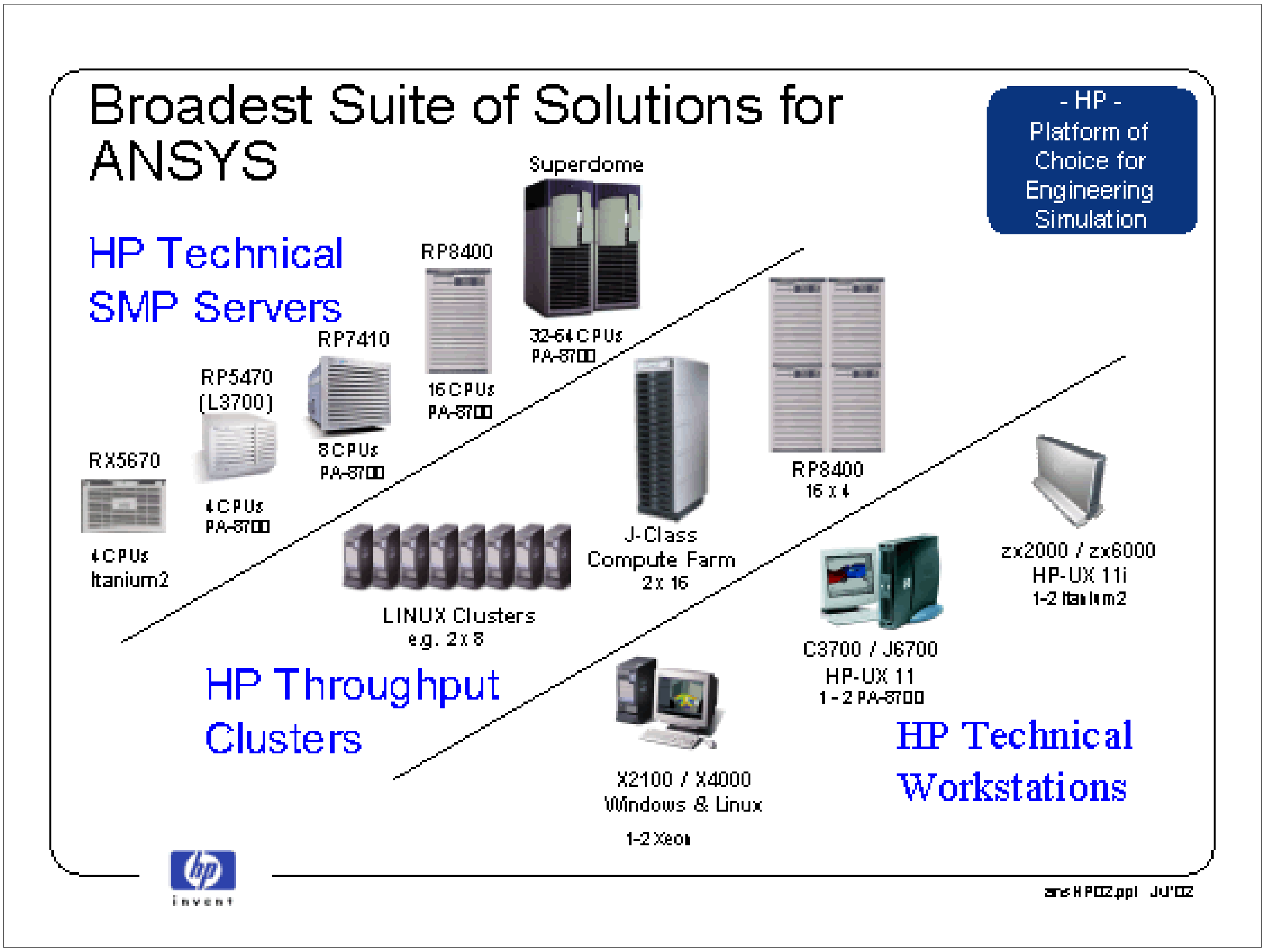


The Focus

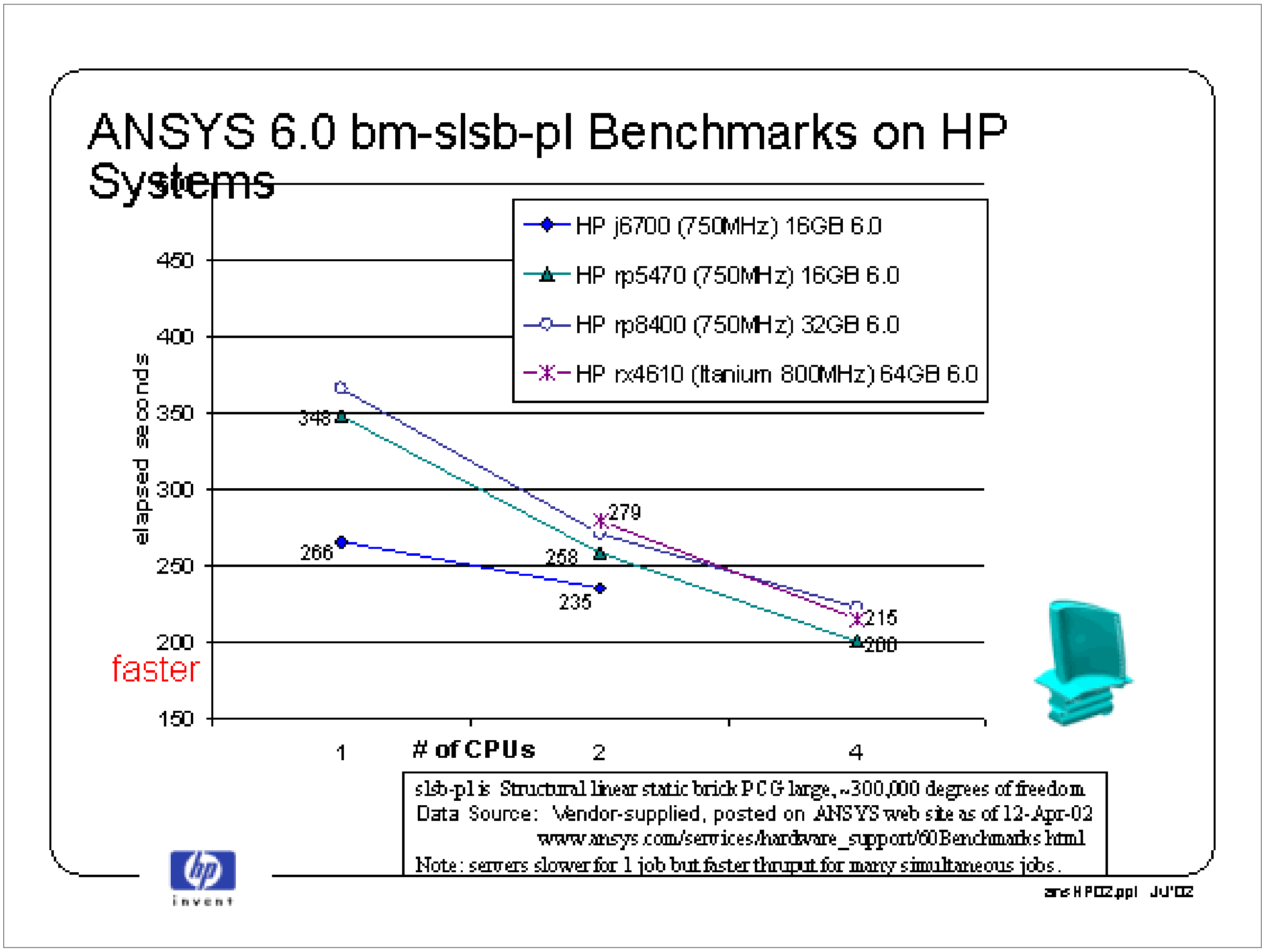
A Publication for ANSYS Users



The Focus
PADT
A Publication for ANSYS Users

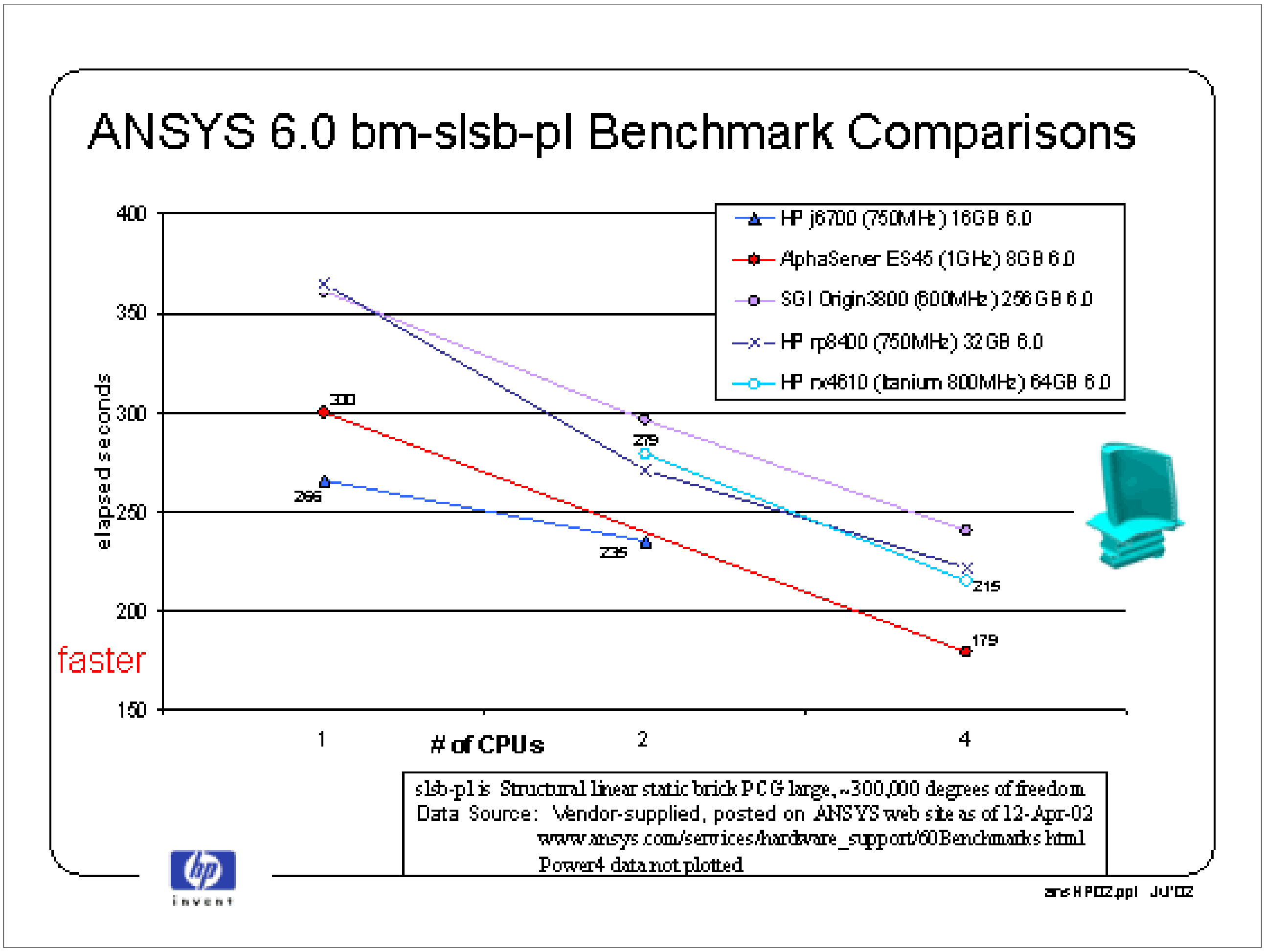


The Focus
PADT
A Publication for ANSYS Users

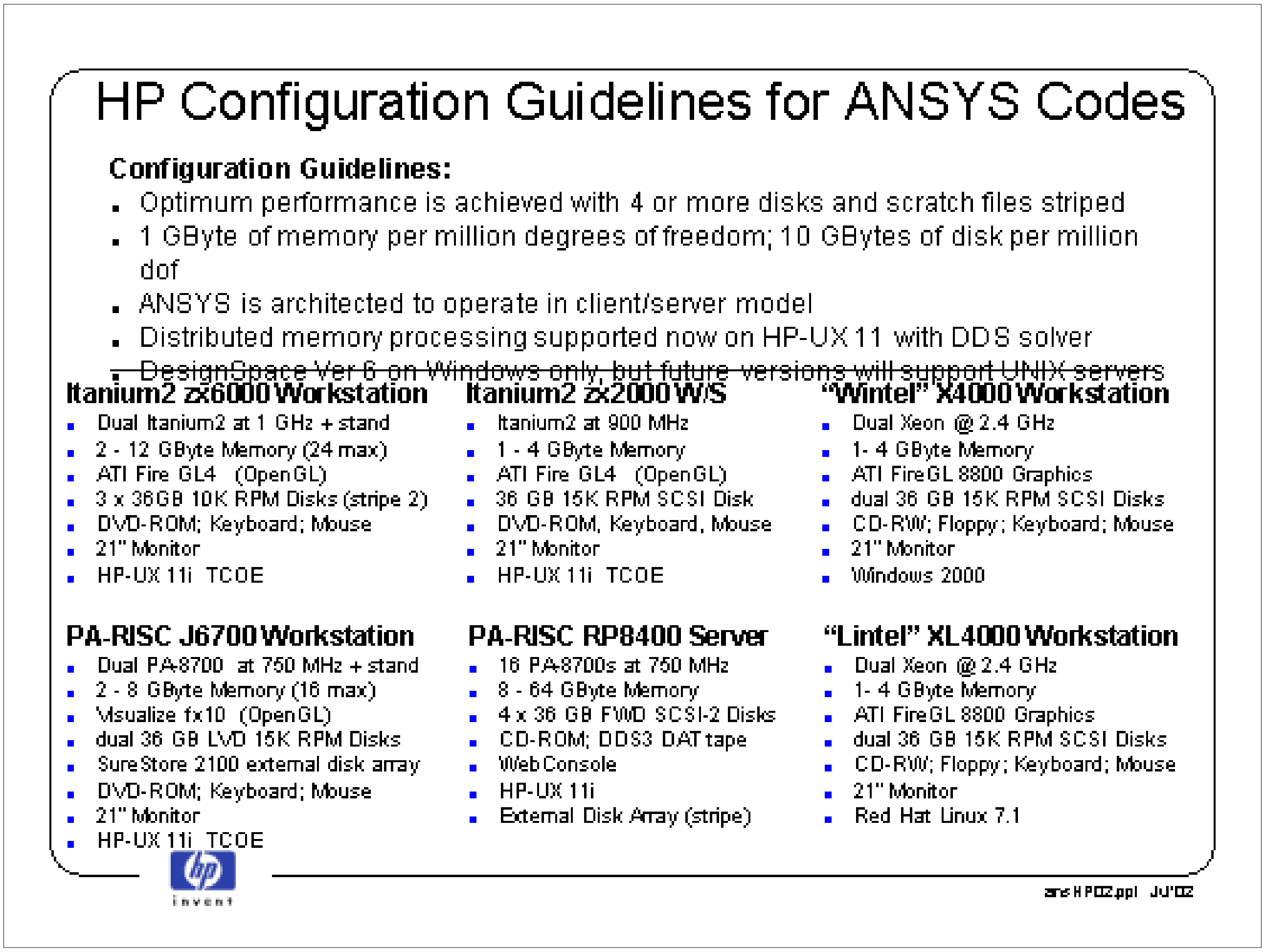


The Focus

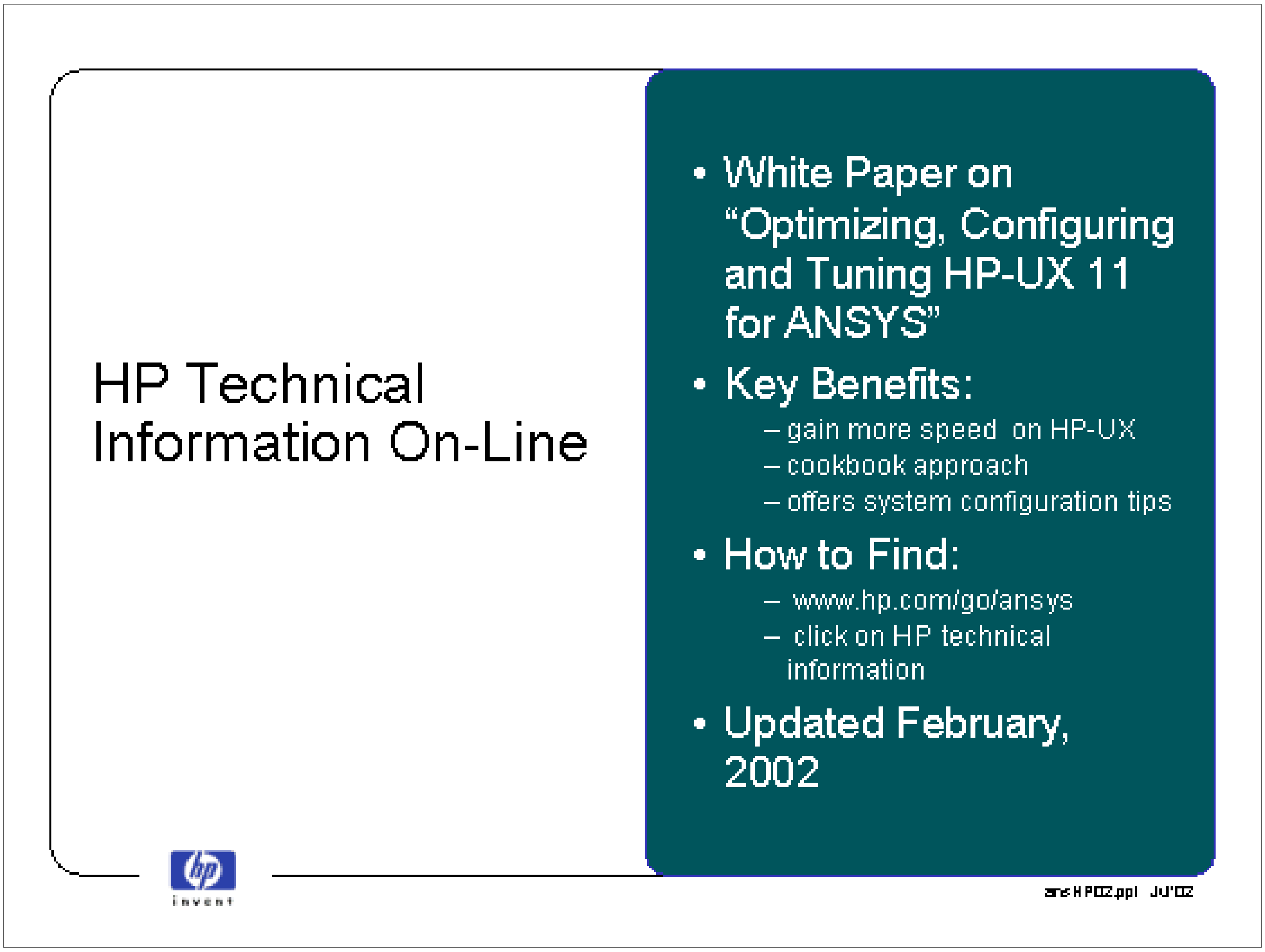
A Publication for ANSYS Users



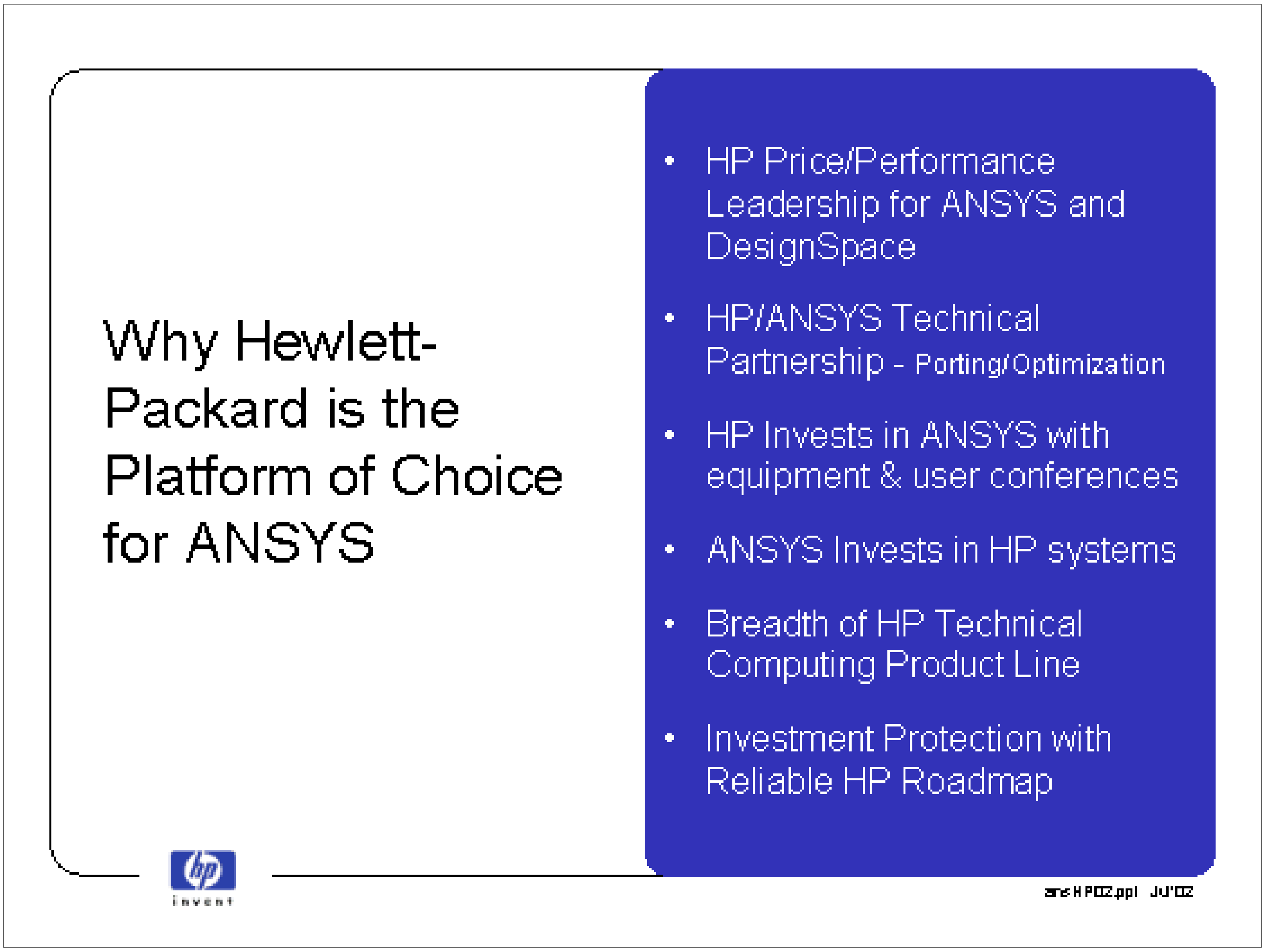
The Focus
A Publication for ANSYS Users



The Focus
PADT
A Publication for ANSYS Users



The Focus
PADT
A Publication for ANSYS Users



The Focus
PADT
A Publication for ANSYS Users

CADfix

CADfix 5.0

- Changes to company
- Changes in ANSYS relationship
- What s new in 5.0

New Name: *TranscenData*

- FEGS in England wrote CADfix
  - Spin-off from their pre/post processor: FEMG
- ITI bought FEGS to get CADfix
- ITI created new company, TranscenData
  - Holding company for their CAD/CAM/CAE products
- Work still being done in England by FEGS guys

New Relationship with ANSYS, Inc.

- Previously, ANSYS resold CADfix and added their own semi-automatic interface
  - CADfix for ANSYS
- Now, there is a Batch Healing Tool based on CADfix
  - ANSYS Geometric Healing Model (ANSYS GHM)
  - Will be the subject of future article in *The Focus*
- Interactive CADfix must be purchased from TrancsenData in the future
- ANSYS GHM does not support IGES
  - A CADfix IGES license is required

The Focus PADT
A Publication for ANSYS Users

New in 5.0

- Overall 50% improvement in geometry processing speed
    - Enhancements to underlying NURB libraries
    - Overhaul of code
    - More efficient and unified data structure
- Improved direct connection to and from CATIA V4
- Enhancements and updates to readers
    - VDA-FS, STEP, ACIS, and PARASOLID
- Support for STEP assemblies
- Assembly Manager tool
- Improved feature removal
    - Automatic hole detection and removal
    - Automated sliver/hole repair based on user defined criteria
- STL file support improved
    - Binary out on STL file
    - More controls on STL output
- Streamlined user interface
- Lots of bug fixes
    - Handles really bad geometry with fewer crashes
    - Fixes to graphics
    - Fixes to GUI
- Company is in transition and attempting to break through
- Future enhancements will be based on large customer demands
- PADT recommends keeping at least one full copy of CADfix

The Focus PADT
A Publication for ANSYS Users

About *The Focus*

The Focus is a periodic electronic publication published by PADT, aimed at the general ANSYS user. The goal of the feature articles is to inform users of the capabilities ANSYS offers and to provide useful tips and hints on using these products more effectively. *The Focus* may be freely redistributed in its entirety. For administrative questions, please contact Rod Scholl at PADT.

The Focus Library

All past issues of *The Focus* are maintained in an online library, which can be searched in a variety of different ways.

Contributor Information

Please don t hesitate to send in a contribution! Articles and information helpful to ANSYS users are very much welcomed and appreciated. We encourage you to send your contributions via e-mail to Rod Scholl.

Subscribe / Unsubscribe

To subscribe to or unsubscribe from *The Focus*, please visit the PADT e-Publication subscriptions management page.

Legal Disclaimer

Phoenix Analysis and Design Technologies (PADT) makes no representations about the suitability of the information contained in these documents and related graphics for any purpose. All such document and related graphics are provided as is without warranty of any kind and are subject to change without notice. The entire risk arising out of their use remains with the recipient. In no event, including inaccurate information, shall PADT be liable for any direct, consequential, incidental, special, punitive or other damages whatsoever (including without limitation, damages for loss of business information), even if PADT has been advised of the possibility of such damages.

The views expressed in *The Focus* are solely those of PADT and are not necessarily those of ANSYS, Inc.