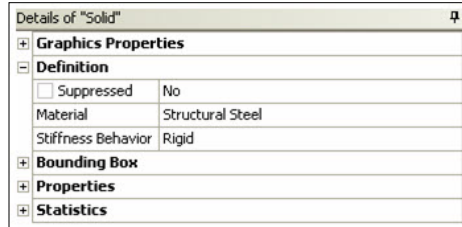


Rigid Surfaces in Workbench

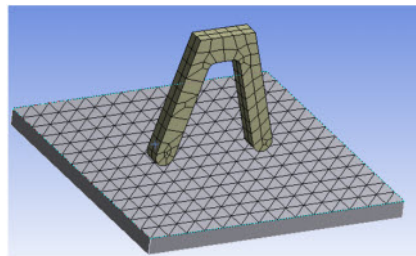
By Doug Oatis

One of the biggest additions to Workbench v11 was the joint utility and rigid body solver. With that inclusion, we now have the ability to perform rigid-to-flexible body contact. For those unaware, this capability exists within ANSYS Classic, and allows you to assume parts as rigid bodies (e.g., metal forming/drawing). You can do this by modeling the rigid part as just a surface (for 3D applications) or a line (for 2D application). Workbench currently only supports rigid to flexible for a 3D analysis (not plane stress/strain or axisymmetric). If you plan on doing this analysis, you should have 'Beta Options' turned on (Tools > Options > Common Settings > User Interface).

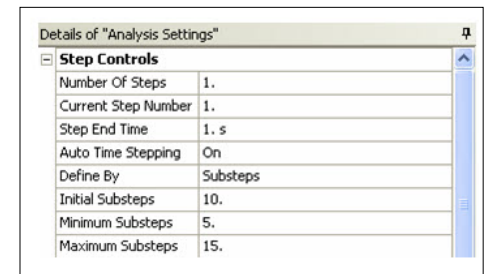
To perform this analysis in v11, you first need to model all of your interacting parts. Once you import the assembly into Simulation, you can then set the 'Stiffness Behavior' in the geometry branch to be "Rigid."



You can then generate your mesh and see that only the surface of your 'Rigid' body has been meshed. You can apply whatever mesh controls to obtain an adequate mesh. You can see from the figure that I applied a mapped mesh control to the top surface of my rigid plate.



In order to move your rigid part, you need to apply a remote displacement. This will serve as a pilot node for the top surface (refer to ANSYS Classic for pilot node information). Make sure the rest of your model is properly constrained, and that you use automatic time stepping to ease convergence.



After the model solves, you may note some oddities in plotting the displacement results for the rigid surface. This happens because of internal contact trimming performed by

(Cont. on pg. 2)

CMS Step 2: Transient Dynamics

By Eric Miller

Still have the same problem? Still have the lone seat of NASTRAN and the crusty "guru" that your company keeps around to run it? In the last meeting about retiring NASTRAN did he say something like "Well, ANSYS does CMS but they don't do transient dynamics with it... we have to have transient dynamics and only NASTRAN can do that!" Too bad crusty "guru" guy, at version 11.0 that statement is no longer true.

If you read the lead article in Issue 43 of *The Focus*, you will find a discussion about using Component Mode Synthesis (CMS)

for modal analysis and, in hindsight, some rather nasty but funny comments about NASTRAN users [link](#). It also gives a pretty good background about what CMS is and how it works in ANSYS and points out that modal analysis was just a first step in development of CMS in ANSYS. The second step came in Version 11.0 with CMS support of static, transient, harmonic and spectral analysis. Of these the most significant in our opinion is full transient dynamics (TD), the solution of a structural problem in the time domain accounting for momentum and acceleration.

The problem with solving TD in ANSYS is that ANSYS is an implicit solver (AUTODYN and LS-DYNA are explicit) so it does a full solve for every time step. That can take a while when you are solving over a second or two. So people generally use CMS in NASTRAN or, if there is mate-

(Cont. on pg. 2)

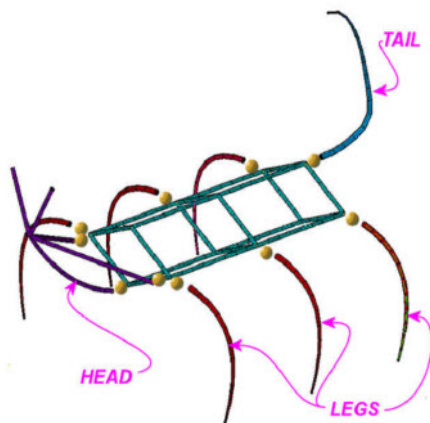


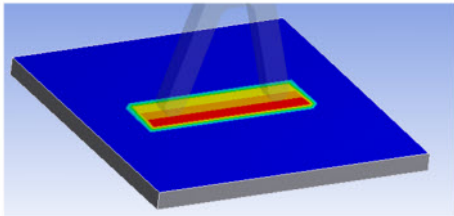
Fig. 1: Model Geometry

Table of Contents

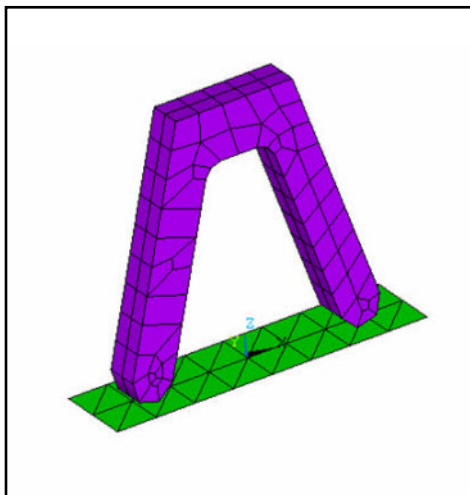
Rigid Surfaces in Workbench	1
CMS Step 2: Transient Dynamics	1
Stressed Penguin: Linux and ANSYS	3
Super Simulation Snippets: RPSD in Workbench.....	4

(Rigid, cont...)

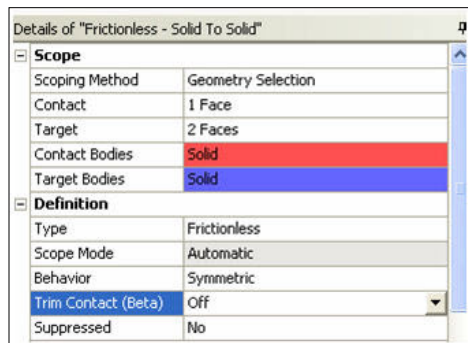
Simulation. Simulation uses contact trimming to reduce the number of contact elements in a model.



You can see the effects of this trimming by writing out an ANSYS input file and viewing the model in ANSYS Classic.



You can change this behavior by setting 'Contact Trimming' to "Off" in the details window for your contact pair.

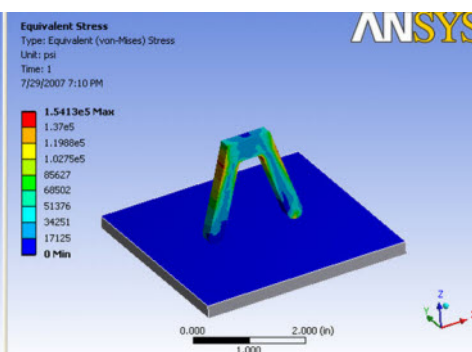
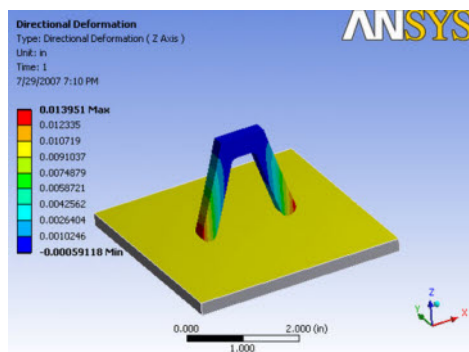


When you do this, the contact region will not be trimmed and all the elements will be

solved for. The results with the modified contact settings are shown below.

This analysis was performed without the ANSYS Kinematics or Rigid Dynamics licenses. If I wanted to specify any type of joint between bodies (revolute, spherical, slot, etc.), I would need those licenses to perform the analysis. You can do this analysis as both a static structural or transient, just make sure your time stepping controls are adequate enough to capture the system response for the transient environment.

Many problems can be simplified with rigid contact. Take some time to read the ANSYS manual on rigid-flexible contact to learn more



(CMS, cont...)

rial nonlinearity, an explicit dynamics code for this type of problem. Now we can use ANSYS to solve those linear transient dynamics problems right in ANSYS.

The process is pretty much the same as for a modal CMS run: 1) build your CMS substructures, 2) solve your problem using the substructures, 3) expand the parts you want at the times you want. The big difference is that in step 2, for the use pass, you do an ANTYPE, TRANS instead of MODAL, and specify the proper time dependent loading for your part. In many cases this is a maneuver load for an aircraft or missile.

Figure 1 shows our silly test case model. All of the macros can be downloaded from here [LINK](#). The non-CMS simulation used RUNFULL.INP and the CMS solve used GENCMS.INP, USECMS.INP and EXPCMS.INP. The CMS model is broken into 4 pieces: the body, head, tail and legs. Comparing run times shows the Full Run taking 2,208 wall clock seconds, and the

CMS run took 139 seconds. That is an order of magnitude different! The difference will go up and down depending on how large your model is, but should still be a huge factor.

The results between the two methods were identical. Figure 2 shows a graph with the deflection of the head and tail as well as the input acceleration load. All values are for the X direction, from tail towards head.

This simple test case shows that ANSYS, Inc. has stepped up and delivered

on the promise of Aerospace transient dynamic simulation. If you get the opportunity to use this powerful and time saving feature, give it a try and let us know how it goes. You can learn a lot more about CMS in the ANSYS manual.

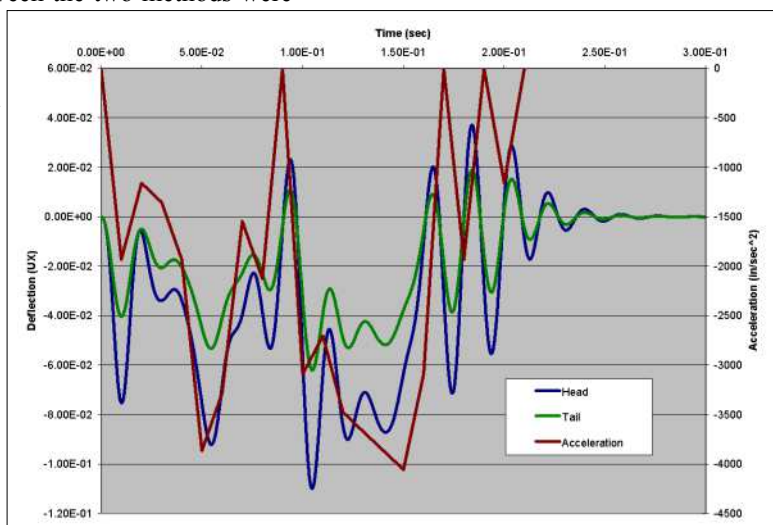
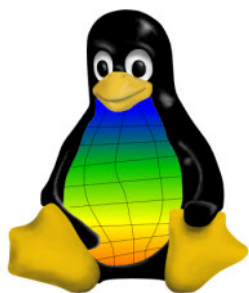


Fig. 2: Input Acceleration and Response Displacement

Stressed Penguin: Linux and ANSYS



By Eric Miller

I recently had to do some real work - I had to actually run an analysis that took a bit of horsepower. Since we recently decided to hide a new Linux compute server from

our CFD guys, I thought I might as well run over there. Now most of my Linux/Unix experience was done a while back so I'm not the most current user out there. What started as a "I'll submit it over there" turned into a couple of hours of updating and re-learning how to work on a remote Linux machine from a windows box. There are a lot of buzzwords here, and I look them up in Wikipedia to figure stuff out, and marked the most important ones with <wl>. So here is some of what I learned:

Linux

We often get the following question: "which Linux should I use?" Well, if you want to play it safe, only use the ones listed in the ANSYS install manual. Currently that is Red Hat 3.0 and SuSe ES9

If you are a risk taker, you can load some of the OpenSource versions of these OS's like CentOS or Fedora. The only thing is that if you want to do that, get ready to download some libraries and fix glitches.

The most important consideration that you can have when deciding to run ANSYS on Linux is that you have a good Linux person on staff or easily reached. For a while we have been recommending Linux compute servers to customers and I have to admit that for many of them it has been a mistake. Linux is cool, efficient, and hip, but it is also different and if you don't know it, you can spend days trying to get things working.

Cygwin

We have mentioned in The Focus before that we like to use Cygwin, the OpenSource Unix and Xwindows server for MS Windows. Go to www.cygwin.com to download the setup program. Be warned, it is a very different setup! You get to choose what modules you want, and there are a lot. If you skip an important one, you are kind

of SOL. If you have the space, install it all. If not, take the defaults then make sure you get everything in Base, Net and X11 <wl>. Some other tips:

1) Make sure you pick a good mirror <wl> to download from. I like mirrors/mcs.anl.gov, it seems robust. But if you have trouble getting the whole install to work, try a different site to download from.

2) Update on a regular basis. Legions of people even nerdier <wl> than FE users are typing away in their mother's basement making improvements to the robustness and speed of this package.

3) You can see your Windows drives through the path /cygdrive/c, /cygdrive/d, etc... Note that most Linux shells are case sensitive.

SSH <wl>

SSH (Secure SHell) is the way to go when connecting to your Linux machines. Even if you have telnet, it is a good habit to use SSH not only for security but also because it handles the XWindows going back to your windows box very well. No need to define displays and permission. Some suggestions:

1) Use -Y as an option when you launch SSH. This tells SSH to tunnel the Xwindows through the HTTP connection, avoiding all that Dispaly nonsense that can be a real pain.

2) Use the -C option. This forces compression and speeds things up. Along the same

lines, -c blowfish speeds up the encryption a little.

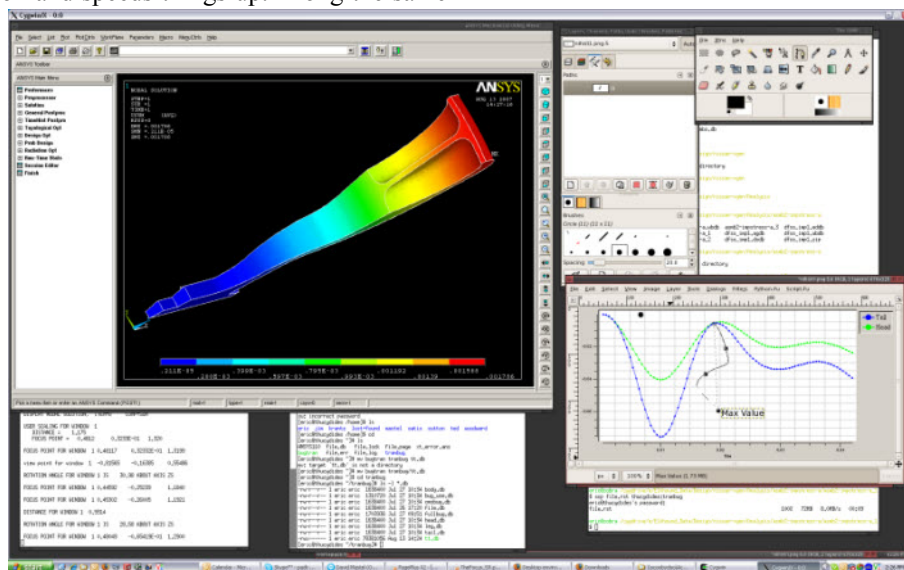
3) If you don't have your Linux drives available for Windows (usually via Samba <wl>), then use the SSH copy tool: scp.

XWindow Desktop

Xwindows is a pretty basic system for displaying graphics and text. In order to take it from that basic level to a system that controls windows and has graphical utilities for productivity you use something called a desktop environment <wl>. They usually have clever artistic names like GNOME or Aqua. On Xwindows, the two most common are KDE and GNOME. These are nice but the main problem with them is that they are big and slow down your computer and worse, your connection. So what you need is a lightweight desktop.

You could use the old default: TWM <wl> but it is very 1992 and not fun (images at the link show just how old it looks). FVWM2 <wl> has a modern look and feel, but is a bit clunky and not so robust. But now there are new options perfect for our use. Someone recommended I try Blackbox <wl>. I'm told Openbox <wl> works well too. The picture below shows my ANSYS model in the Blackbox desktop. Even the 3D rotation works pretty well.

That is about it, hopefully this will save some of you some time and pain while you figure out how to run on Linux, and grow to enjoy being outside the Windows world.



Super Simulation Snippets

RPSD Curves from a PSD Run

As a variation on our usual Awesome APDL feature, we want to mix in some APDL scripts that you can use to expand the capabilities of Workbench Simulation. And since we have an unnatural urge to always give things silly names: we will call this feature Super Simulation Snippets.

We will start with one that we get a lot of requests for. In V11 of Simulation, you can do a PSD run very easily. And much of the typical output is readily available. Moreover, if you want a response or RPSD curve, you can't get that automatically.

Enter APDL and code snippets to once again save the day. This snippet, provided by our intrepid snippeteer Doug Oatis, is a good example of how to include POST26 post processing into a Simulation model.

```
! Enter Post26 and store PSD info
/post26
store,psd,5
! Grab mass node and base node
! You need to define d_base as a named selection
esel,s,ename,,21
nsle
*get,mass_n,node,0,num,max
allsel,all
cmsel,s,d_base
*get,base_n,node,0,num,max
allsel,all
! Get response info
nsol,2,mass_n,u,y
nsol,3,base_n,u,y
rpsd,6,2,,3,1
rpsd,7,3,,3,1
! Multiply to get the right units
prod,2,6,,,mass_node,,,1/386.4**2,1,1
prod,5,7,,,mass_node,,,1/386.4**2,1,1
! Set up your graph options
/axlab,x,Frequency (Hz)
/axlab,y,PSD (G**2/Hz)
/gthk,axis,2
/grtyp,0
/gropt,ascal,on
```

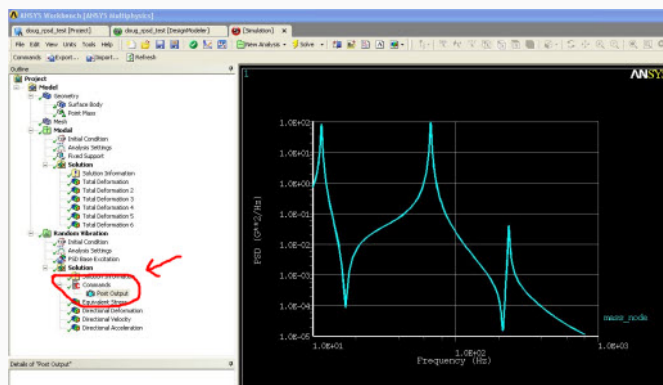
```
/gropt,logx,on
/gropt,logy,on
/xrange,default
/yrange,default,,1
! Plot it (vars 2 and 3)
/show,png
/gfile,600
plvar,2,3
/show,close
! Write to a file for external plotting
*get,n_data,vari,1,nsets
*dim,psd_data,array,n_data,5,1
vget,psd_data(1,1,1),1 !frequency data goes here
vget,psd_data(1,2,1),2 !mass1 data here
vget,psd_data(1,5,1),5 !base node data here

*cfdopen,'d_test','txt'

*vwrite,'Freq','mass_n',' ',' ','d_base'
%14C %14C %14C %14C %14C

*vwrite,psd_data(1,1,1),psd_data(1,2,1),psd_data(1,3,1),
psd_data(1,4,1),psd_data(1,5,1)
%14.5G %14.5G %14.5G %14.5G %14.5G

*cfclose
allsel,all
/post1
```



News - Links - Info

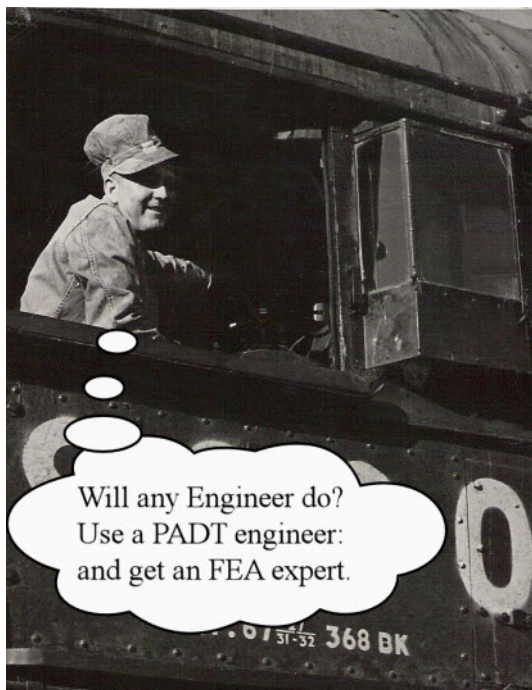
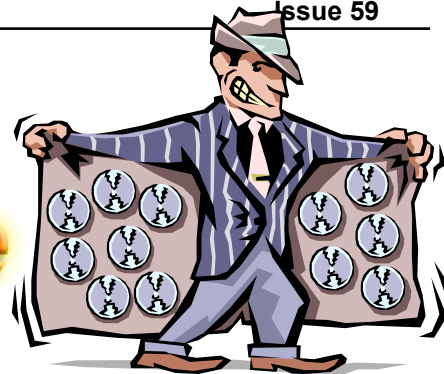
- ANSYS Posts Another Record Quarter and Increases 2007 Estimates [<link>](#)
- ANSYS Recognized as 2nd Fastest Growing Software Company in US, and only CAE Company on List. [<link>](#)
- ANSYS-FLUENT-CADFEM Meeting in Germany: Combining the European ANSYS-FLUENT User Conference with the 25th CADFEM User Meeting Nov 21-23rd in Dresden, Germany. CADFEM throws the best (technical and fun) meetings in the world. If you can make it, do so! [<link>](#)

Upcoming Training Classes

Month	Start	End	#	Title	Location
Aug '07	8/22	8/24	402	ANSYS High Frequency Emag	Tempe, AZ
	8/30	8/31	701	Design Optimization and Prob.	Tempe, AZ
Sep '07	9/6	9/7	301	Heat Transfer	Tempe, AZ
	9/10	9/12	101	Intro to ANSYS, Part I	Albq, NM
	9/13	9/14	102	Intro to ANSYS, Part II	Albq, NM
	9/17	9/20	802	Advanced APDL & Customization	Las Vegas
	9/24	9/25	201	Basic Structural Nonlinearities	Tempe, AZ
	9/26	9/27	204	Advanced Contact and Fasteners	Tempe, AZ
Oct '07	10/1	10/2	104	ANSYS Workbench Sim. Intro	Tempe, AZ
	10/4	10/5	203	Dynamics	Tempe, AZ
	10/8	10/9	302	ANSYS WB Sim. Heat Transfer	Tempe, AZ
	10/15	10/16	100	Eng. with Finite Element Analysis	Tempe, AZ
	10/18	10/19	105	ANSYS WB Sim Struct Nonlin	Tempe, AZ
	10/18	10/19	107	ANSYS WB DesignModeler	Las Vegas
	10/24	10/26	902	Multiphysics Simulation for MEMS	Tempe, AZ
	10/30	10/30	411	Workbench Simulation for EMAG	Tempe, AZ

The Focus is a periodic publication of Phoenix Analysis & Design Technologies (PADT). Its goal is to educate and entertain the worldwide ANSYS user community. More information on this publication can be found at: <http://www.padtinc.com/epubs/focus/about>

The Shameless Advertising Page



Electronic Mail is a term that's been bandied about data processing circles for years. Simply put, it means high-speed information transportation. One of the most advanced methods is terminals talking to one another.

Your mailbox is the terminal on your desk. Punch a key and today's correspondence and messages are displayed instantly.

Need to notify people immediately of a last-breaking development? Have your messages delivered to their terminal mailboxes electronically across the hall or around the world.

Electronic Mail is document distribution that's more timely, accurate and flexible than traditional methods.

There's no mountain of paperwork.

Administrative personnel are more effective.

Managers have access to more up-to-date information. Decision-making is easier.

Tomorrow's automated office will clearly include Electronic Mail. But like the rest of the Office of the Future, it's available at Honeywell today.

For more information call Mr. Laurie Reeves at (800) 293-3229/3 (within the 617 area, call 552-5048). Or write him at Honeywell Office Automation Systems, Three Newton Executive Park Drive, Newton Lower Falls, Massachusetts 02469.

Honeywell

What the heck is Electronic Mail?



Kiva.org is a non-profit revolutionizing the fight against global poverty by enabling people to connect with and make personal loans to low-income entrepreneurs in the developing world.

Through Kiva, anyone can make a direct loan of as little as \$25—and in doing so contribute to a vital influx of capital to the developing world where most of the poor are self-employed Entrepreneurs.

Innovative: "Innovation of the Week: Kiva.org uses smart designs to make a little cash go a long way."

ChitMoney: "If you've got 25 bucks, a PC and a PayPal account, you've now got the ingredients to be an international financier."

Join PADT in making a real difference in people's lives by simply making a small loan.



\$500 loaned so far
\$0 needed
\$25

