

Best practices for efficient HPC performance with large models

Dr. Hößl Bernhard, CADFEM (Austria) GmbH

PRACE Autumn School 2013 - Industry Oriented HPC Simulations, September 21-27, University of Ljubljana, Faculty of Mechanical Engineering, Ljubljana, Slovenia













Agenda

- " Part I:
 - . Prerequisites
 - " MPI. HP or Intel
 - " Hyperthreading and ANSYS
 - . How to Set up HPC
 - " The Remote Solve Manager (RSM)
 - "Solver Handler, Number of CPUs, SMP, and DMP
 - " ANSYS Mechanical APDL
 - . Handling of Large Models
 - " CAD Import
 - " Named Selections
 - " Tree Filter and Tags
 - " Object Generator

Agenda

- " Part II:
 - . Mesh Controls for Large Models
 - " Curvature and Proximity
 - . How to Cut Analysis Time
 - " Contact Settings
 - " Analysis Settings
 - . Evaluation of Results
 - "Results Tracker
 - " Newton Raphson Residuals and RSM
 - " Postprocessing Command Snippets
 - " Max Tag Adjust to Visible

Part I

Prerequisites: MPI. HP or Intel



WIKIPEDIA The Free Encyclopedia

Main page Contents Featured content Current events Random article Donate to Wikipedia

Interaction
 Help



Message Passing Interface

From Wikipedia, the free encyclopedia

Message Passing Interface (MPI) is a standardized and portable message-passing system designed by a group of researchers from academia and industry to function on a wide variety of parallel computers. The standard defines the syntax and semantics of a core of library routines useful to a wide range of users writing portable message-passing programs in Fortran 77 or the C programming language. There are several well-tested and efficient implementations of MPI, including some that are free or in the public domain. These fostered the development of a parallel software industry, and there encouraged development of portable and scalable large-scale parallel applications.

Contents [hide]

Prerequisites: MPI. HP or Intel

- IBM Platform (HP) MPI is a little more robust than Intel MPI.
- "if both can solve, it seems Intel MPI is a little faster (1% or 2%) than Platform MPI.



Prerequisites: Hyperthreading and ANSYS

- Ansys doesn't do any distinction between physical processors (e.g. 2x Xeon processors on two separate sockets) and virtual processors (i.e. cores), and between virtual processors (cores) and logical processors (if you have Hyperthreading enabled, this would "double" the cores you have).
- "This might cause some troubles -> next slide

Prerequisites: Hyperthreading and ANSYS

E.g. running a job on 8 cores with activated hyperthreading would results in the following warning during the solution:

*** WARNING *** CP = 0.499 TIME= 17:04:40

The requested number of shared-memory processors (8) exceeds the number of physical processors that are available (4). As the use of virtual processors is not recommended, the number of processors used will be 4.



Why RSM?



- Client: Computer, used for Pre- and Postprocessing
- *Job:* Simulation
- Compute Server: these are the machines on which jobs are run
- *RSM* (Remote Solve Manager, aka: RSM Host): dispatches jobs to computing resources
- *Queue:* A queue is a list of Compute Servers available to run jobs

- *Efficient use of licenses*
 - . Single license: one license can be used for all tasks at a time

. **Split license:** split of the license in Pre-Post and Solve-Batch. During solution the Pre-Post license can be used to prepare the next simulation or for postprocessing.



["] Login as Administrator and change Dir to install folder ANSYS



Install Manager and/or Server service



" Start RSM:

ANSYS 14.5: Start > Programme > ANSYS 145 > Remote Solve Manager

	🔥 ANSYS® Rei	mote Solve M	lanager 1	L 4.5					<u> </u>
RSM Manager	File Options	View Tools	s Help						
Queues	E - & My Comput E - Ø Queues E - Ø Loca In E - Ø Comput	er I bcalhost e Servers host	Job	Status	Submi	Owner	Priority	Server	Queue
	wie-hulk-de	esk							.::

" Setup RSM:

ANSYS® Remote Solve Manager 14.5							
File Options View Tools	s Help						
All Jobs 🔹 🛸 🎎 👸	2						
🖃 🔩 My Computer	Job	Status	Submi	Owner	Priority	Server	Queue
E-S Local							
localhost							
E Compute Servers							
localh Proper	localh Properties						
Delete.							
Test Se	erver						.::

RMB

" Setup RSM:

ľ	Compute Server Propert	ies		×	
	General Cluster SSH				
I	Display Name	wie-beh-lap			
	Machine Name	wie-beh-lap]	
	Working Directory Location	User Specified	▼		
	Working Directory	D:\temp_RSM		1	
	<< Less				
	Server Can Accent Jobs	~	User must have all	rig	hts for this Dir
	Maximum Number of Jobs	1 🕂			
	Limit Times for Job Submi	ssion			
	Start Time 00:00:00		_		
	End Time 00:00:00		v		
	□ Save Job Logs to File				
	Delete Job Files in Working	g Directory			
			OK Cancel <u>H</u> elp		16

" Setup RSM:

Compute Server Properties	×
General Cluster SH	
Cluster Type None	
Shared Cluster Windows HPC	
File Managemer	isk
More >>	Define Cluster Type
L	
	OK Cancel <u>H</u> elp

" Set your Password (your login)



" Test the Server



" Test the Server



" Add a RSM manager

🔥 ANSYS® Rem	ote So	olve M	anager 14	4.5								
File Options	View	Tools	Help									
All Jobs	- 83	Y D	esktop Ale	rt	1							
🕞 💑 My Computer		🛸 R	emove	Entf			Status	Submitt	ed	Owner		
		S	ubmit a Jo	b	ting w ⁱ	hah lan	Finishad	21.00.2	010 10.04.0	CADEE		
E-V Local		C	ntions			Λ Optic	ons					×
E	servers	3				Solve M	anagers:					
🛛 🔤 🛶 🥡 wie-be	eh-lap					🔽 localł	nost			C Desktop A	lert Settings -	
🥻 🦾 💑 wie-hulk-desk	<									Show	Running Jobs	
										Show	Pending Jobs	
ļ									1	Show	Completed Jol	bs
						Name:	wie-hu	k-desk				
					~	Add	De	elete	Change		OK	Cancel

Add a RSM managerSet your Password (your login)

🔥 ANSYS® Remote Solve Ma	nager 14.5		_ [
File Options View Tools	Help			
All Jobs 🔹 🛸 🛤 🔁				
📄 🧔 Local 📃	Job	Status	Submitted	Owne
Compute Servers	Ø wie-beh-lap/Testing wie-hulk-desk	Finished	21.09.2013 13:33:34	CADF
e & wie-hulk-dool				
Queue 🕼 Properties				_
🗄 🤕 Loc 👭 Accounts				
🗄 🧭 Compl 🍪 Set Passwo				
wie-nak-aesk				•
				.::

" Add a RSM manager

🔥 ANSYS® Remote Solve Ma	nager 14.5			
File Options View Tools	Help			
🛛 All Jobs 🔹 🛸 🛤 💆	-			
🛛 🖂 Local 📃	Job	Status	Submitted	Owner
Compute Servers	•			Þ
				-
E Compute Servers				
Pro	perties			•
🏓 Del	ete			.::
Tes	t Server			

Add a RSM manager



The following ports must be open for Communication between RSM componentsANSYS 14:8140 and 9140ANSYS 14.5:8145 and 9145

How to Set up HPC . Solver Handler

" Setup solver handler

🕅 B : Static Structur	al - Mechanical [ANSYS Multipl	nysics]			
File Edit View Units	Tools Help		🥥 ++:		Define Solve Manager	
🛛 🐨 ht 🤹 🖒 🕶 🖒	🔚 Write Input Fi	le	€ €		Set Queue	
四	Read Result F	iles	Suppr		Set License	
🛛 🗹 Show Vertices 🖓।	Solve Process	Settings				J
Environment 🍳 Inertia	Addins		onditic		\square	
Outline	Ø Options				/	
Filter: Name 🔻	Variable Mana	Solve Process	Settings			×
Project	Run Macro	Figener Co	moutor	Add Local		
📄 🙆 Model (B4)		Eigener Co	mputer Hint	Add Local	Computer Settings	
H		wie-beh-lap	3 CPUs		Solve Manager: wie-hulk-desk	
		wie-hulk-de	sk_8_CPUs	_Add Remote _	Queue: Local]
				Cat an Default	Liconco: ANSYS Mechanical	7
				Set as Default		-
				Rename	Advanced	.
				Delete	OK Cancel	

How to Set up HPC. Solver Handler

" Setup solver handler

	Solve Process Settings		X
DMP	Eigener Computer Eigener Computer, Hint wie-beh-lap_3_CPUs wie-hulk-desk_8_CPUs	Add Local Add Remote	Computer Settings Solve Manager: wie-hulk-desk Queue: Local
Π		Set as Default	License: ANSYS Mechanical
Advai d Propert	ies	×	Advanced
🗆 Distribute Solutio	on (if possible)		OK Cancel
Use GPU acceleratio	n (if possible) None	-	
Max number of utiliz	zed processors: 8		
Manually specify	Mechanical APDL solver memory	J₀s #CPU	Js
V	Vorkspace: 0	MB	
	Database; 0	MB	
Additional Command	d Line Arguments:		

How to Set up HPC. Solver Handler

" Use the desired solve handler

🔀 B : Static Structural - Mechanical [ANSYS Multiphysics]	
File Edit View Units Tools Help 🔮 ↔	🔰 Solve 🔻 ?√ Show Errors ?√ Review Mod
📽 🚧 💽 🖌 🗞 🕏 💽 💽 🖉 🗸 😔 🗨	Eigener Computer
J 🖆 🚽 🚽 🖓 Selection ▼ 💡 Visibility ▼ 🔽 Suppres	Eigener Computer, Hintergrund
」 ダ Show Vertices 🍄 Wireframe 🛛 🛄 Edge Coloring マ	wie-beh-lap_3_CPUs
Environment 🔍 Inertial 🔻 🍳 Loads 🔻 🛠 Supports 👻 Conditions	
Outline	⁴
Filter: Name 🕶 🔯 🕢 🕀	Static Structural
Project	Time: 1, s
🗄 🖷 🞯 Model (B4)	21.09.2013 13:47
⊕ ~~ √ Geometry	A Joint Condition Swing: 0, °
E Coordinate Systems	B Joint Condition Boom: 0, mm
E √ Connections	G Joint Condition Arm: 0, mm
Mesh	D Joint Condition Bucket: 0, mm
Static Structural (B5)	E Remote Displacement

The solver handler are saved in the file "solvehandlers.xml" which can be located in %appdata%\ansys\programversion.

How to Set up HPC. Mechanical APDL

" Start ANSYS product launcher

▲ 14.5: ANSYS M	echanical APDL Product Launcher [Profile: *** La	ast ANSYS Run ***]	Hostname: wie-beh-lap	<u> </u>
File Profiles C	Options Tools Links Help			
Sim	ulation Environment:	Ad	d-on Modules	
	SYS 🗾 🔽		S-DYNA (-DYN)	
	ense:		NSYS DesignXplorer (-DVT)	
ANS	SYS Multiphysics			
File Management	/ Customizatio / High Performance Preferences Computing Setup			
Type of High Per	formance Computing (TIPC) Run.			
O None				
 Use Share 	d-Memory Parallel (SMP) Number of Process	ors 2 🛛 🗆 Use C	GPU Accelerator Capability	
 Use Distrit 	buted Computing (MPP)			
MPI Type	*			
				-
O Use	e Local Machine Only Number of Processors			
O Spe				
	<u>^</u>	^	New Host	
	Add ->		Remove	
	~		Edit	
0.115	MPI File		Browco	
	(absolute pa	ath required)	Blowse	
	RunCancel	Run	Product Help	
	- Concern		roadericip	

Handling of Large Models . CAD Import

- " Import/Attach feature now supports two options for target geometry type:
 - . DesignModeler
 - Full conversion to DMc internal representation (Parasolid)
 - Traditional behavior
 - Workbench
 - ["] Light-weight B-rep
 - ⁷ Faster import from external CAD system into DM
 - > DM automatically converts entities during modeling, as needed
 - > Manual conversion with the new £onversionqFeature

De	etails View	P
	Details of Import1	
	Import	Import1
	Source	C:\Users\userdata\sk\dm\\cyl_head.prt
	Target Geometry Type	Workbench 💌
	Base Plane	Workbench
	Operation	DesignModeler
	Direction in the second s	· · · · · · · · · · · · · · · · · · ·

110 minutes	5 Minutor
14.0 ((WBgeometry
Import Time in	Import Time in
Conversion	

- " Named selections can used to:
 - . Filter items
 - . Group items
 - . Convert items
- " This is of interest for:
 - . Programming macros
 - . Fast selection of similar items used for Preand Postprocessing
 - . Easier navigation through the model

"How to insert a Named Selection item #1



"How to insert a Named Selection item #2



" Insert Toolbar for fast selection

🕅 B : Sta	tic Structural - Mechanical [ANSYS Multip	ohysics]
File Edit	View Units Tools Help	🔄 🥑 📑 🛛 🤣 Solve 🔻 ?
] 😨 🎠 *	 Shaded Exterior and Edges 	🕂 Q, Q, Q, Q, Q, Q,
	Shaded Exterior	1 - 1 - 1 - 1 - 1x
Named Se	Wireframe	
Outline	Graphics Options	₽
Filter: Na	 Cross Section Solids (Geometry) 	
Projec	Thick Shells and Beams	
	 Visual Expansion 	
	Ruler	
±	Legend	
Ē∕¶	Froded Nodes	
	Large Vertex Contours	
	Display Edge Direction	
	Annotation Preferences	
	Outline	
	Toolbars	Named Selections
	Windows •	Unit Conversion
		 Graphics Options
	····√φ Joint Condition Arm	✓ Tree Filter
	Solution (B6)	Joint Configure
I –		

"Working with the toolbar

📔 A : Static Structural - Mechanical [ANSYS Multiphysics]			
] File Edit View Units Tools Help	▼ ?、		
🖫 🎶 K T 🔖 TK K K 💽 🚭 T S 💠 Q Q Q 🔍	Q		
mounts 🔹 🖶 Selection 👻 🖓 Visibility 🔻 🕼 Suppression 💌			
보 또 Show Vertices 🛱 Wireframe 🛛 📕 Edge 💡 Hide Bodies in Group	1		
Named Selection			
Outline Show Only Bodies in Group			
Filter: Name 🔻 😰 🖉 🛨 2	1.09		
Project			
🖮 🎯 Model (A4)	m		
⊡…, 🏟 Geometry			
E			
Connections			
Resh			

→ Example: excavator.*wbpz; helicopter.wbpz*

Handling of Large Models . Tree Filter and Tags

- " Tree Filter:
 - . Users want the ability to control the amount of data shown in the Mechanical tree view.
 - . The Tree Filter allows users to show only those objects which match some search terms.
 - . The filter searches on *object name* or on *object tags*.

Handling of Large Models . Tree Filter and Tags

"Entering a search term, e.g. ‰olt,+and hitting Enter filters the tree to show only those objects with names matching ‰olt.+




A similar search can be performed on object tags by selecting \$\overline{Tags+from the drop-down menu.}



 Multiple search terms are supported, so all objects with %Jate+and %Jasket+can be found with %Jasket plate + (case sensitive).



When filtering is in effect, any object that is added, removed, or renamed will not be affected until filtering is manually refreshed.



["] The Tree Filter can be turned off under View \rightarrow Toolbars (Mechanical recalls last visibility setting on startup).



→ Example: *tree_filter.wbpj*

- " Object Generator
 - . Users want the ability to make multiple copies of objects, scoping each to different geometry.
 - . The Object Generator allows users to use an existing object as a template for replication.
 - . Almost any tree object that supports ⁽Duplicate+can be used as a template.

- *Typical Application:*
 - . The user has many bolts in a model and wants to apply a bolt pretension load to each bolt



- [″] Step 1:
 - . Define the tree object to be copied (optionally define Details).



- [″] Step 2:
 - . Select the geometry to which the tree object should be copied.



- ["] Step 3:
 - . Open the Object Generator and click Generate+



- " Result:
 - . The original load was copied to all selected geometry. *All details* from the original template object are maintained.



- Task:
 - . Connection with many bolts, axial forces in the bolts needed
- *Idea*:
 - . Use Beams for easy postprocessing
- " Drawback:
 - . Multiple preprocessing actions needed, 1 bolt pretension per bolt and 2 connections for each of the beams
- " Solution
 - . Use object generator for convenience.

→ Example: *object_generator.wbpz*



- " Preparation:
 - . We need **named selections** for the object generator. If the pattern operation is done in **Design Modeler** you can define the component before creating the pattern and use propagate selection, so you only need to click one face.



-	Details of Koppelflae	tails of Koppelflaeche		
	Named Selection	Koppelflaeche		
	Geometry	22 Faces		
	Propagate Selection	Yes		
	Export Selection	Yes		

- " In ANSYS Mechanical
 - . Define the object you want to copy multiple times.





• - * •

- In ANSYS Mechanical
 - . Activate the object generator.
 - . Assign the named selections
 - . Define a tolerance
 - . Specify a prefix, so you can easily see the generated connections
 - . As a result you receive multiple joints.



- Now we can repeat the process with the bolt pretension.
 - . Define first bolt pretension
 - Use named selections with all line bodies (tolerance not needed, because there is a clear relation)
 - . Generate pretension for all 11 bolts.



Part II

Mesh Controls for Large Models -Curvature and Proximity

The proximity size function allows you to specify the minimum number of element layers created in regions that constitute 'gaps'.





Mesh Controls for Large Models -Curvature and Proximity

The curvature size function examines curvature on edges and faces and computes element sizes on these entities such that the size will not violate the maximum size or the curvature normal angle.





curvature normal angle

Mesh Controls for Large Models -Curvature and Proximity

Important: using curvature and/or proximity usually results in very big models because small details are densely meshed.

-> HPC might be important

["] The key concept of HPC is domain decomposition

- . Break problem into N pieces (domains)
- . "Solve" the global problem independently within each domain
- . Communicate information across the boundaries as necessary



- When using contacts each of the contact pairs has to be analyzed in the same domain.
- ["] Large contact areas slow down the analysis.
- "Help: trim contact areas
- " APDL command: CNCHECK,TRIM

" Example of socket with non-linear contacts

Meshing results (contact elements only)



" Example of socket with non-linear contacts

Performance Analysis: ANSYS 12, INTEL Nehalem CPUs + Infiniband interconnect



- ["] D-ANSYS (DMP) vs. ANSYS (SMP)
 - . The entire SOLVE phase is parallel: including element matrix generation, linear equation solver, and element results calculation
 - . More of the analysis is performed in parallel
 - . Better scaling than SMP, between 2X to 7X speedup on 2 to 16 processors!
 - . Memory usage also scales in most cases.
 - Disk (I/O) usage also scales. Separate I/O files are managed on each processor

- Sparse Solver (Direct Solver)
 - . Efficient when the resulting stiffness matrix is **unsymmetric** (e.g. irreversible operations like contact with friction, plasticity)
 - Efficient when the resulting stiffness matrix is ill conditioned (e.g. weak springs, numerous contacts, plasticity with K->0, skewed elements)
 - Can be used in contact analysis with Lagrange
 Formulation
 - Can be used for **mixed u-P element Formulation** (hyperelasticity, plasticity)

- " PCG (Iterative Solver)
 - . Efficient with **very fine** meshes of solid elements
 - . Working with symmetric matrices only
 - . Takes less RAM than Sparse Solver
 - . Swaps between RAM and Disk -> SSD-Disks no problem
 - . When using the command PCGOPT it can be used when **joints** (MPC184 elements) are present in the model

- " Solver options that save analysis time
 - . bcsoption,, incore for Sparse Solver
 - . dspoption,,incore,,,,performance for DSparse Solver
 - . pcgopt, lev_diff: settings for level of
 difficulty
 - " when using up to 4 cores stick with AUTO
 - "When using 8 or more cores then:
 - If AUTO picks a Lev_Diff value = 1, stick with that
 - If AUTO picks a Lev_Diff value ≥ 2, consider lowering Lev_Diff value by 1

Evaluation of Results – The Result Tracker

["] Check solution status during run using RSM



Evaluation of Results – The Result Tracker

Insert result item you like to track during solution – has to be done before analysis

🖻 B : Static Structural - Mechanical [ANSYS I				
File Edit View Units Tools Help				
📽 ht 🧚 🙀 k 🕶 🖏 🔻 k k k k 💽 💽 🗳 🕶				
J 🖆 🔹 👻 Selection ▼ 💡 Vis				
토 또 Show Vertices 🔐 Wireframe 🛛 🛄 Edge Cq 🤈				
] Solution Information 🛗 Result Tracker 🔻 💈 🥵 💙				
Outline				
Filter: Name 🔻 😰 🕢 🕀				
Project				
🗄 🖷 🎯 Model (B4)				
🗄 🗸 🌆 Geometry				
🗄 🥠 🔆 Coordinate Systems				
⊡ √® Connections				
🗄 🗸 🕸 Mesh				
🕀 🐨 🕸 Named Selections				
🚊 🌮 🖉 Static Structural (B5)				
Analysis Settings				
📄 🦗 Solution (B6)				
Solution Information				



Evaluation of Results – The Result Tracker

" Define Details



F	Definition		
L	Туре	Number Contacting	
L	Suppressed	No	
_			
Ρ	scope		
Contact Region Frictionless - plate steel To gasket		el To gasket 🔄	
Ь	Doculto		
	Results	1	
	Minimum		
	Maximum		

What are Newton Raphson Residuals

 $[K^{T}]{\Delta u} = {F^{ext}} - {F^{int}}$



Choose number of iterations for which you like to see the Newton Raphson Residuals



- In order to evaluate the Newton Raphson Residuals in case an analysis is not converging do the following:
- " Open the file:
 - . C:\Program Files\ANSYS Inc\v145\RSM\ Config\xml\Ansys_Generic.xml
- Add the line:
 - . <file type="binary"
 special="SolutionInformation">file.nr*</
 file>

```
<2xml version="1.0"2>
<jobTemplate>
       <script>GenericJobCode.xml</script>
       <debug>FALSE</debug>
       <cleanup>TRUE</cleanup>
       <inputs>
   <file type="ascii" special="cancel">file.abt</file>
 </inputs>
       <outputs>
   <!--<file type="ascii">*.out</file> Use this to get additional *.out files -->
   <file type="ascii" special="SolutionInformation">solve.out</file>
   <file type="ascii" special="SolutionInformation">file.gst</file>
                <file type="ascii" special="SolutionInformation">file.nlh</file>
   <file type="ascii" special="SolutionInformation">file0.gst</file>
   <file type="ascii" special="SolutionInformation">file0.nlh</file>
   <file type="binary" special="PostDuringSolve">file.rcn</file>
   <file type="ascii" special="PostDuringSolve">file0.nlh</file>
   <file type="ascii" special="cancel">file*.err</file>
   <file type="ascii" special="cancel">solve.out</file>
    <file type="binary" special="SolutionInformation">file.nr*</file>
 </outputs>
</jobTemplate>
```

Evaluation of Results – Postprocessing Command Snippets

["] During Postprocessing you can use your own APDL routines to list items, plot results, and do your own calculations.



Evaluation of Results – Postprocessing Command Snippets

Check setting "Invalidate Solution"


Evaluation of Results – Postprocessing Command Snippets

Check setting "Save MAPDL db" -> pre-used variables and Named Selections can be used

Outline	ф.
È √Ê Static Structural (A5) ▲	
Analysis Settings	
🔤 Figure	
√®↓ Standard Earth Gravity	
↓ Fixed Support	
····√♀, Fixed Support 2	
Commands (APDL)	
Details of "Analysis Settings"	
🖻 Analysis Data Management 🔹 🔺	
Solver Files Directory	D:\users\ub\aktuelle
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	Yes 🔽
Delete Unneeded Files	Yes
Nonlinear Solution	No

! Plot eines Ergebnisses	
! white background:	
/RGB, INDEX , 100, 100, 100, 0	
/RGB.INDEX. 80, 80, 80, 13	
/PCP TNDEX 60 60 60 14	
/KGB,INDEX, 00, 00, 00,14	
/RGB, INDEX, 0, 0, 0, 15	
resume,file,db !MAPDL-db einlesen	
set,last <u>!</u> Let zten Datensatz einles	
/SHOW, PNG ! Gibt alle folgenden Plot	
! Bild-Einstellungen machen (Select:	
esel,s,ename,,185,187 <u>! nur Solids</u>	
/TITLE, Element Solution Equivalent S	
plesol,s,eqv ! Element Solution - Un	
/SHOW, CLOSE ! Leert buffer und sch	
ALLS	

Evaluation of Results - Max Tag Adjust to Visible

["] For large assemblies

