Recent developments in OASYS Primer

October 2004

Miles Thornton

- Engineering consultancy – 7000 staff worldwide
- “Arup” is our founder’s name, Ove Arup
- “Oasys” = Ove Arup Systems
**Aims of Oasys software**

- 100% compatible with LS-DYNA – data is never lost or corrupted
- LS-DYNA-specific – no other solver is supported, all efforts go into optimum working with LS-DYNA
- LS-DYNA-expert – our knowledge and experience built into software. The software should understand and work with the complexity of LS-DYNA.
  - Example: more than 2000 error checks are included, based on our knowledge of LS-DYNA.
- Even complex tasks should be quick and error-free – specialist tools for each job
- Responsive to customer requests:
  - Fix any bugs quickly
  - Add users requests into new versions
  - We use a database system to log and track these requests
- REDUCE ERRORS AND WASTED TIME

**New in Version 9.1 Beta 1**

- Faster and better graphics
  - Typically 4-5 times faster in Shaded mode
  - Transparency, colour, plotting mode can be set for all entities
- Redesigned User Interface
  - More intuitive, less mouse movement
  - Quick-pick and short-cut keys
- More access to Part data
  - Part Tree – hierarchical view of model; new Assembly capability
  - Table for viewing/editing part data
  - Quick-pick options for viewing data and accessing editing menus
- New capability
  - Rigidify function and rigid patch creation
  - Cross-references viewer
  - Spotwelder improvements (and auto-welding)
  - (find attached improvements)
  - (finding how a model has been modified)
  - Various small enhancements
User Interface

New screen layout with larger graphics window
Windows-like drop-down menus for less-commonly-used functions

Quick-pick control (more on this in later slides)
Access to Manual (Beta version: only V9.0 manual available)

Tidy button explained in later slides

Back/forward through previous views (e.g. used to go back to a zoomed-in view)

View control and drawing commands (LI/HI/SH/CT etc)

LOCK stores the current blanking status; ALL restores blanking to the status that existed at the time LOCK was pressed. REV reverses blanking.

Tools
Most menus appear here so they never cover the graphics ("docked" menus)
Tabs control which menu is visible (click the tab to see the menu). This removes the need to dismiss menus, and allows return to previous menus without having them covering the graphics area. Up to 6 menus are kept in this way, plus Model and Part Tree are always present. When the 7th menu is called up, the least recently used of the other 6 automatically disappears.

Example: press MAT….
... a new tab appears, with the material menu, by default in MODIFY mode.

Picking control changes automatically to "pick material", so the material to be modified can be screen-picked.

To return to the MODEL menu, click the MODEL tab.
Tabbing and Shortcut keys

- Use the tab key to move through text boxes within a menu
- New shortcut keys: buttons on keyboard that access commonly used functions. Not case sensitive.
  - `?` = list of available shortcut keys
  - `ESC` = dismiss the menu that the mouse is over
  - `RETURN` (or middle mouse button) = APPLY
  - `1,2,3,4,5,6,7,8` = XY, XZ, etc standard views
  - `A` = Autoscale
  - `B` = blanking menu; `R` = reverse all blanking; `U` = unblank all
  - `E` = entity visibility menu
  - `H`, `L` and `S` = perform Hidden line, Line and Shaded plots
  - `M` = measure node-to-node
  - `Z` = zoom (drag across rectangular area)
  - `+` and `=` = zoom in and out
  - `T` = tidy all floating menus; `C` = close all floating menus

QUICK-PICK
Select action, select entity type, then click single items or drag across area
Middle mouse UNDOES quick-pick actions
Right mouse (clicked when over an entity) gives menu of quick-pick actions for that entity
Quick-pick can be used to set plotting mode (shaded, wireframe, etc).

Hint: wireframe images often look best with this option (this menu is accessed from Display menu in top bar, then Options).
Quick-pick “Only” means “blank everything else except the picked item”

Quick-pick “Information” brings up a text box for the picked item, showing label, title, which INCLUDE file the item is in, and other data depending on entity type. For Parts, the information includes thickness, yield stress, element formulation and mass.
Quick-pick “Edit” brings up the standard editing menu for the picked item. In the example here, the entity type is set to Material.

When the entity type is set to a type of element (e.g. Shell), quick-pick “Element details” brings up this panel.
To return to quick-pick mode (e.g. after using Measure and picking nodes), either drop down here to Quick-pick…

… or click the cross

… similarly, picking nodes for Measure can be re-activated by clicking this cross

Entity visibility control

This panel replaces the old VIS1 and VIS2.

Access this menu either by Shortcut E on keyboard

Or

Top bar menu Display then Entities

Or

From viewing/drawing panel, press Ent

Select Category

Draw/label all entity types in category…

…Or, change the draw/label settings for each entity type separately
"Iconisation" of menus

Menus collapse to the right-hand end of the top bar. This is to allow screen-picking etc.

All floating menus can be iconised to top-left of screen, restored to their previous positions, or closed using TIDY options.
Alternatively, press I on the keyboard (shortcut for “Iconise All”)…

… all floating menus are now iconised. Press I again to restore them.
Refresh of Object Menus

Primer can have many menus open at the same time. This could potentially lead to confusion if, for example, a part were deleted in one menu but then still appeared in the “object menu” (list of items for selecting) in another menu. These lists are now automatically refreshed in most cases where data is created, edited or deleted.

A new button enables manual refreshing of the object menu where necessary.

These menus automatically grow wider when the mouse moves in, so the titles can be seen more fully.

Summary of methods of blanking

• Shortcut keys U (unblank all), R (reverse blanking) and B (blanking menu)
• Quick-pick (drag across area or click on each item)
• Lock and All buttons within viewing panel allow storing of blanking status (e.g. to view a subset of the model)
• Blank, Unblank and Only in Part tree, then click on parts, include files or assemblies
• Right click on parts, include files or assemblies in Part Tree
• Part Table, tick Blanking column under View
• Can also switch entity types on/off (shortcut E or Display Entities)
• Can also switch models on/off (Model tab, then List)
Summary of methods to find which INCLUDE file an item is in

• If you can see the item on the screen,
  – Use Quick-pick “Information”, click on the item
  – Alternatively, if the item is a part, use Quick-pick “Locate in Tree” to see it in the Part Tree under its Include file
• If you know the name or ID of a part
  – Part Tree “Find”
• KEYWORD => PART => LIST – text box contains INCLUDE file information
• From Element Details panel
  – Press INCL
• Or,
  – Put the item on the clipboard, right-click and LOCATE

Find attached

Demonstration
Find attached - Introduction

Attached works in a slightly different manner to previous versions of Primer. It is now separate from the entity visibility so it is possible to find one type of entity attached without affecting the visibility of other types of entity.

The default state of the attached menu is set to find any structural connections.

The new menu offers several extra options:

- **Tied contacts** switch allows attached to find elements through tied contacts.
- **Recursive** switch will loop through attached until nothing more can be found.

Options are available to find a whole part or step through single elements for both rigid and deformable parts in one iteration.

Find attached - Default

Pressing Apply in the default mode will find any structural connections in the model. Tied contacts are found by default and use routines from the Primer contact penetration checker.

Other entities can also be found by turning their switches on in the panel:

Several iterations
Find attached – Whole parts

Whole part option allows the user to skip the element-by-element iterations, and find a whole part instead.

Find attached - Recursive

Recursive option will loop through attached iterations until Primer cannot find any more entities.

Reversing the model display (shortcut R) will display any unattached entities.
The Part Tree is a new facility allowing viewing of the model structure and access to blanking and other functions. By default, the tree view is organised by Include files, then Parts.

Select then drag and drop (or multiple-select with shift and/or CTRL, then right-click and cut/paste) to move parts to a different Include File. This performs the same action as Clipboard MOVE TO INCLUDE with Find Referenced Items – nodes and elements are moved as well as Part, Section and Material data.
Part tree

While dragging, the cursor symbol changes when a valid “drop point” is reached. The parts will be put into the Include file that is under the cursor when the mouse button is released.

Part tree – blanking

1. Select blanking function
2. Click on Include files or Parts (shift-click or CTRL-click for multiple selection)
**Part tree – blanking**

ONLY displays only the selected items (in this case, an INCLUDE file)

Can also right-click from part tree to set transparency etc.
Part Tree “Find”

“Find” searches by name or Part ID

If you don’t know the name of a part but can identify it on the screen, use Quick-pick option “Locate”, pick the part, then the Part Tree expands to show the picked part. Sub-option ONLY picks one part at a time; sub-option ADD allows multiple parts to be selected in the tree.
Assemblies

Assemblies are user-defined groupings of parts, that can be used for blanking or selecting for other operations.

The two main difference compared with Sets or Groups are:

Assemblies can be hierarchical – assemblies can contain assemblies.

No part can be defined in more than one assembly

This tree structure can be created in the Part Tree menu: right-click on the model or existing assembly to create a "child" assembly. This can then be renamed.

Parts can be dragged into each assembly (or cut and pasted). Example: unblank only the parts required; use quick-pick “Locate Add” to highlight the parts in the Part Tree; right-click cut; on the assembly, right-click paste.
Assemblies

The assembly structure can be written to and read from a separate file. This method is used to allow a single definition of the assembly hierarchy to be kept centrally and used for many models.

When the keyword file is written, a comment is written with each *PART to indicate which assembly the part is in. However, the keyword file does not contain the assembly hierarchy.

Assemblies

The assemblies are then available for selection in other menus, e.g. ORIENT.
Cross-Reference Viewer

Demonstration

To start the cross-reference viewer, EITHER use Xrefs button in Tools panel, then select item from object menu…
… or press VIEW_XREFS from edit panels

This side shows entities that refer to the selected item. In this example, a part was selected; all keywords containing a reference to that part are shown.

This side shows entities that are referred to by the selected entity, e.g. *PART refers to a material ID and a section ID.
Expand a category to show individual entities of that type; click on an entity to see references to that entity. Here, Part 238 is referred to by Part Set 1, which is referred to by Contact 2. The left side of the view works “up the tree”, while the right side works “down the tree” – both starting from the picked item.

To see both up and down the tree from one of the entities displayed (e.g. Contact 2), right-click the entity and select “Xrefs”…
The references to and from Contact 2 are now displayed.

Return to the original view (references to the picked part) by using the **tabs**.
Part table shows information for each part in the model.

Window can be any size. Column width and order is adjustable. Sorting can be done using any column.

<table>
<thead>
<tr>
<th>Part ID</th>
<th>Part ID</th>
<th>Part ID</th>
<th>Part ID</th>
<th>Part ID</th>
<th>Part ID</th>
<th>Part ID</th>
<th>Part ID</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
</tr>
<tr>
<td>SHELL</td>
<td>SHELL</td>
<td>SHELL</td>
<td>SHELL</td>
<td>SHELL</td>
<td>SHELL</td>
<td>SHELL</td>
<td>SHELL</td>
</tr>
<tr>
<td>01</td>
<td>02</td>
<td>03</td>
<td>04</td>
<td>05</td>
<td>06</td>
<td>07</td>
<td>08</td>
</tr>
<tr>
<td>09</td>
<td>10</td>
<td>11</td>
<td>12</td>
<td>13</td>
<td>14</td>
<td>15</td>
<td>16</td>
</tr>
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<td>18</td>
<td>19</td>
<td>20</td>
<td>21</td>
<td>22</td>
<td>23</td>
<td>24</td>
</tr>
<tr>
<td>25</td>
<td>26</td>
<td>27</td>
<td>28</td>
<td>29</td>
<td>30</td>
<td>31</td>
<td>32</td>
</tr>
</tbody>
</table>

Many items available for each part, e.g. element formulation can easily be added.
Information can easily be changed. 

e.g. to change element formulation from 2 to 16 for some parts

<table>
<thead>
<tr>
<th>Part ID</th>
<th>Part Name</th>
<th>Section ID</th>
<th>Mat ID</th>
<th>Element Formulation</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>lower_steel</td>
<td>81014</td>
<td>70139</td>
<td>16</td>
</tr>
<tr>
<td>3</td>
<td>steel_tophat</td>
<td>15</td>
<td>153001</td>
<td>16</td>
</tr>
<tr>
<td>5</td>
<td>sill_swim_me</td>
<td>3</td>
<td>220100</td>
<td>5</td>
</tr>
<tr>
<td>27</td>
<td>SHELL</td>
<td>81014</td>
<td>70139</td>
<td>16</td>
</tr>
<tr>
<td>28</td>
<td>lower_steel</td>
<td>20</td>
<td>005000</td>
<td>9</td>
</tr>
<tr>
<td>32</td>
<td>SHELL</td>
<td>25</td>
<td>050000</td>
<td>14</td>
</tr>
<tr>
<td>33</td>
<td>SHELL</td>
<td>25</td>
<td>050000</td>
<td>14</td>
</tr>
<tr>
<td>34</td>
<td>upper_steel</td>
<td>26</td>
<td>000000</td>
<td>16</td>
</tr>
<tr>
<td>35</td>
<td>lower_steel</td>
<td>26</td>
<td>000000</td>
<td>16</td>
</tr>
<tr>
<td>36</td>
<td>steel_tophat</td>
<td>40</td>
<td>153001</td>
<td>16</td>
</tr>
<tr>
<td>60</td>
<td>SHELL</td>
<td>26</td>
<td>000000</td>
<td>17</td>
</tr>
<tr>
<td>61</td>
<td>lower_steel</td>
<td>45</td>
<td>000000</td>
<td>23</td>
</tr>
<tr>
<td>65</td>
<td>SHELL</td>
<td>50</td>
<td>050000</td>
<td>28</td>
</tr>
<tr>
<td>66</td>
<td>SHELL</td>
<td>50</td>
<td>050000</td>
<td>28</td>
</tr>
<tr>
<td>101</td>
<td>front_support</td>
<td>3</td>
<td>220100</td>
<td>5</td>
</tr>
</tbody>
</table>

Select the rows you want to change and type in the new element formulation number.
Items that will be updated are shown in red.

In this example a new section card is created as not all of the parts using section 3 were selected.

Apply will save the changes.
Spotwelding

Demonstration

Selecting Spotwelds – new options

Attached to panels selects all welds that are attached to any of the parts (panels) selected.
Selecting Spotwelds – new options

Multiple seams selects those welds that are attached only to selected parts (panels). This allows one or several weld seams to be chosen.

Single seam selects only those welds that are attached to ALL the parts (panels) selected – e.g. if three panels are selected, only spotwelds connecting all three panels will be shown. This allows identification of a spotweld seam.
Spotwelding

Check panel will now also check that spotweld nodes are sufficiently close to the panel mid-planes to stick to the panels. In some cases, the test used may be more demanding than that used by LS-DYNA. If any spotwelds fail the test, the fixing panel appears.

Spotwelding

After reprojecting or checking welds, the “fix” panel is displayed for welds that need attention.
Spotwelding

Fixing panel now has more options:

Show entire weld seam shows chosen bad weld with red crosshair, other bad welds with white crosshair.

Show connected beams shows any spotwelds connected to the same panels.
Fixing panel now has more options:

- **Restore blanking** sets blanking to stored status
- **Store blanking** allows user to set blanking status as displayed

Option to save bad welds to file when exiting, so they will not be lost when the panel is dismissed.

File to save welds to
Weld errors have more explanation than in previous versions of Primer.

Spotwelds and Part Replace

Part Replace now allows automatic reprojecting of attached spotwelds.
Spotwelds still stick after Shell PID change

If the Part ID of shell elements is changed (e.g. creation of a rigid patch), the PID references on the spotweld beams (ELEMENT_BEAM_PID) are now automatically changed to ensure that the spotwelds still stick to the same elements.

Autowelding – introduction

New Auto weld function in Primer intelligently finds potential weld points between selected shells and then attempts to weld at all of the calculated points.

Weld runs are determined by flagging shells that are close together with similar normal vectors, and then using the model free edges on these flagged shells as “weld runs”
Autowelding – explanation

- Length of 4 blue part sides = “weld run length”
- “Sub run length”
- “Sub break angle”
- Weld run start
- Weld run finish
- Weld pitch
- Weld edge distance

Autowelding – user parameters

Primer allows the user to adjust several parameters to determine how the weld runs and points are calculated:

- **Min run length** – all weld run lengths found must be greater than this length
- **Min sub length** – weld runs are split up using the angle between each edge, the sub lengths must all be greater than this value
- **Sub break angle** – determines the angle at which weld runs are broken up into sub lengths
- **Pitch** and **Weld edge dist** – tell Primer how far along, and how far in from the edge run to weld
Autowelding – Sketch positions

Sketch weld positions will sketch all the potential weld points Primer has calculated to attempt to weld. A green line marks the start of a run, and a red line marks the end.

Autowelding – parameters

By adjusting the sub break angle from 30 to 50 degrees, the two sub runs displayed on the left can be combined into one, resulting in a better distribution of weld points.
The red dots are the successful 2-panel welds, and the green dots are the 3-panel welds. Even though there were weld runs (and therefore weld points) that were next to each other, the auto weld routine checks for proximity and will not weld any points that are too close to each other.

Extra panels can easily be auto-welded in a model with current welds. The only step needed here was to define the shells to weld between the grey and cyan part.
Here, the red panel has a weld run that has stopped short of the edge because the Sub break angle is too small.

The green panel has no potential points because the Min run length is too large.

**Autowelding – parameters**

By adjusting the parameters to the settings below left, Primer can find good auto-weld points without the user needing to manually create the welds.
“Rigidify” function

Rigidify - creating rigid patches

Pick elements for each rigid patch and press Apply

Move or copy elements to new rigid part
Rigidify - creating rigid patches

Selected elements moved into new rigid parts and RB merge

New parts labelled at highest+1 in include file

Rigidify - Making part of a model rigid

Selected elements will be moved into new rigid parts created starting at user defined label

Select elements by screen area
Selected elements have been moved into new rigid parts. Thickness, density and elastic modulus are copied from original parts. Rigid body merges are created to join the new rigid parts together. The master rigid body is a new “dummy” part that does not itself have any elements.

On completion, delete function automatically called to remove invalidated constraints.
Autowelding – potential

Primer can also weld an entire car in one go, although we don’t recommend this as good practice!

Mesh independent airbag folding

Demonstration
Traditional folding process

- Time taken to work out fold pattern on initial flat shape
- Time taken to mesh, including constant-width "tram-lines"
- Need to start again if fold pattern is changed.
- Takes several hours

Mesh-independent folding process

EITHER define size of circular airbag OR start with any arbitrary mesh that defines the outline shape

Primer re-defines the mesh appropriate to each fold. The "tramlines" parallel to each fold are automatically generated. Reference geometry is automatically updated to the new mesh.
Changing a fold

Delete unwanted fold...

Create new fold. Mesh is automatically changed.

Set DT2MS

CONTROL >> Calc DT2MS
Enter timestep and % added mass is calcd
Or
Enter % added mass and timestep is calcd
Press SET DT2MS to set the timestep
**64bit versions**

32bit executables limited to approx 2Gb.

Manipulating huge models may require more memory. 64bit executables give this.

64 bit executables available for:
- HP PA-RISC
- HP Itanium
- SGI Irix
- IBM AIX
- SUN Solaris

Windows 64bit: awaiting official release of OS.
Linux 64bit: under development

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**Version 9.1 Beta 1 available now**

Contact DYNAmore for details

**Version 9.1 available in November**
Recent developments in OASYS Primer

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