Using ANSYS with Remote Solve Manager at Windows

Prerequisites: Configure PuTTY SSH

Step-by-step guide

1. Install the same version of ANSYS as that at cluster (2020 R2 for now) at your Windows machine.
2. Login to the cluster (say Fidis) and add the following line to the .bashrc file.
   ```bash
   export AWP_ROOT202=/ssoft/spack/external/ansys/2020R2/v202
   ```
3. At your Windows machine, launch the RSM Configuration application (only need to do the steps 3-5 once), for example, select Start > ANSYS 2020 R2 > RSM Configuration 2020 R2
4. Define RSM Configurations.
   - Click + to create a new RSM configuration for the cluster and give the configuration a name (e.g., the name of the cluster)
   - Specify information on HPC Resource, File management, and Queues as shown below
   - In the field HPC type, select SLURM from the drop-down menu (directly integrated for ANSYS 2020 R2 and later)
   - In the field labeled SLURM job submission arguments (optional), enter (for example to request wall time of 1h and mem-per-node of 120GB): `--time=1:00:00 --mem=120G`
   - Select Use SSH communication to the submit host and enter your username at cluster.
   
   Note: Submit host depends on the machine you use. The values for maximum time will need to be changed once you begin submitting actual jobs.
4. Note:
To test the new configuration, click on Submit.

5. Quit the RSM Configuration application.
6. Launch the ANSYS Workbench, for example, select Start > ANSYS 2020 R2 > Workbench 2020 R2
7. Create/Open your workbench project file (.wbpj).

Submitting a Mechanical Job from Workbench to Remote Solve Manager:
- On the Project Schematic, double-click either the Model or the Setup cell to launch Mechanical.
- In the Mechanical application, on the Home tab, select Solve Process Settings.
- In the Solve Process Settings dialog box, click Add Queue.
- In the Rename Solve Process Settings dialog box, enter a Solve Process Setting Name (for example, Cluster), then click OK.
- In the Solve Process Settings dialog box: In the left pane, select the solve process setting whose name you just specified.
- In the Settings pane, select the RSM Queue to which the Mechanical job will be submitted.
- From the License drop box, select the ANSYS product license to be used for the solution (ANSYS Academic Research Mechanical and CFD).
- In the Advanced Properties dialog box: Select the Distribute Solution (if possible) option. Specify the Max number of utilized cores. Click OK.
- In the Solve Process Settings dialog box, click OK. The dialog box closes and the solve process setup is complete.
- Select the Solve drop-down on the toolbar. You will see the solve process name you just defined (in this example, Cluster). Select that process.
- Tick Distributed and specify Number of Cores (Note: However, the 2020R2 version ignores this and uses the maximum cores per node: 36 for Helvetios and 28 for Fidis to run the job).
- Click Solve. The solve commences. When the solution has completed, the Solution branch and the items underneath it in the project tree will each have a down arrow next to them.
- Right-click Solution and select Get Results.

Submitting a Fluent Job from Workbench to Remote Solve Manager:
- In the Fluent system, right-click the Solution cell and select Properties.
In the Solution Properties view, Clear **Use Setup Launcher Settings** and Set Solution Process properties as follows:

- Right-click the Solution cell and select **Update** or select **Update Project** on the toolbar.

Submitting a Design Point Update from Workbench to Remote Solve Manager:

- Right-click the **Solution** and select **Properties** to set the **Update Option** property to **Run in Foreground**.
- Right-click the **Parameter Set** and select **Properties** to set the **Update Option** property to **Submit to Remote Solve Manager (Legacy)**.
- From the **RSM Queue** drop box, select the queue (note: i.e. partition in slurm) that will be used for the job.
- Set the **Job Submission to One Job for Each Design Point**.
- For **Component Execution Mode**, specify **Serial** or **Parallel** solver execution mode. The **Parallel** option is available only if the selected solver supports parallel execution mode.
- For the parallel option, set the **Number of Processes** (i.e. the number of tasks per job) equal to or less than the maximum CPUs per node (36 for Helvetios and 28 for fidis).
- Use the defaults for the other settings.
- Proceed with updating the project or design points. For example, select **Update Project** or **Update All Design Points** on the toolbar.

**Related articles**

- Using the clusters
- Using ANSYS with Remote Solve Manager at Windows
- How to configure PuTTY SSH to transfer files to Cluster from Windows
- How to use Jupyter and Tensorflow on Izar
- Cryosparc on Izar