

# Using ANSYS with Remote Solve Manager



New users of ANSYS should get access to the license first by going to this page: <https://ansys.epfl.ch/> and following the instructions.

## Step-by-step guide

1. Login to the cluster (say Fidis) and load the ANSYS module.

```
module purge
module load ansys
export LD_LIBRARY_PATH=/ssoft/spack/external/ansys/2020R2/v202/Framework/bin/Linux64/Mesa
```

2. Launch the RSM Configuration application (**only need to do the steps 2-4 once, recommend VcXsrv as Windows X Server**).

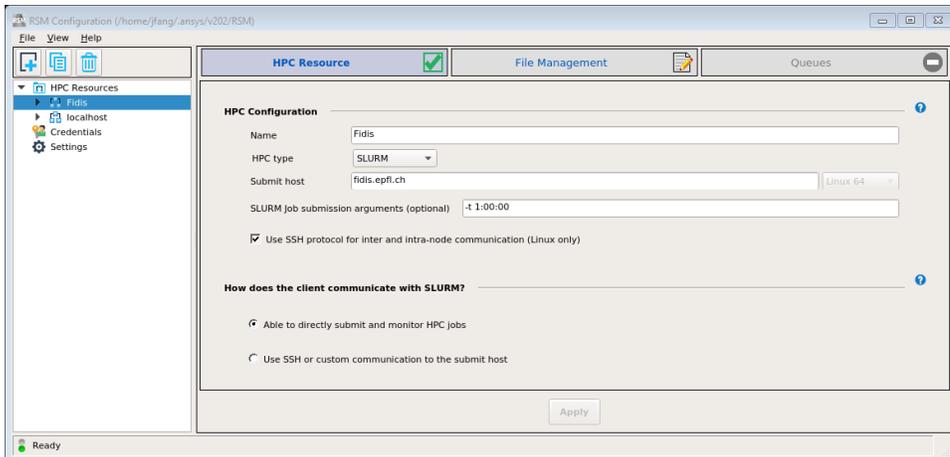
```
rsmclusterconfig
```

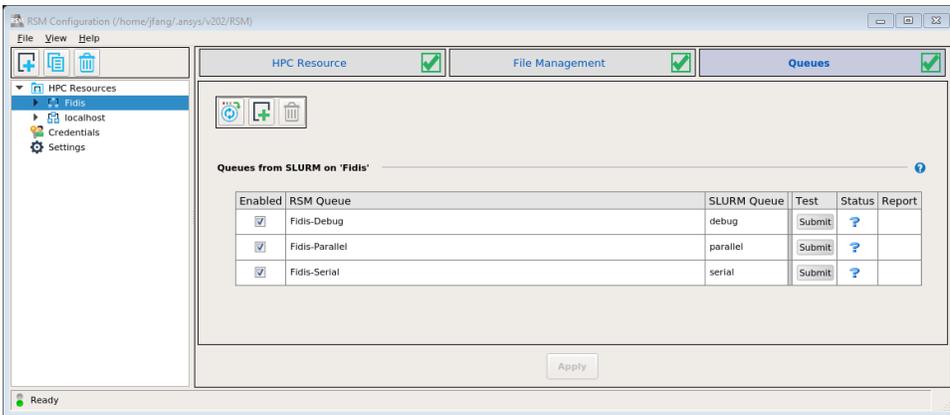
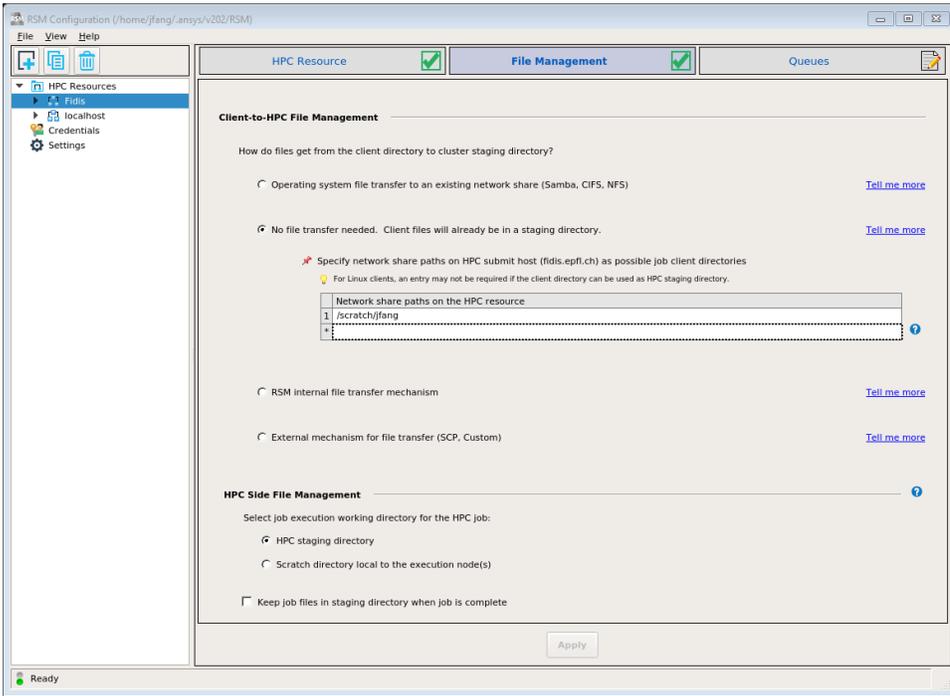
3. Define RSM Configurations.

- 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: **-t 1:00:00**

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.*





**Note:** SLURM Queue name **serial** is at the moment of writing this document only available on Fidis. To test the new configuration, click on Submit.

4. Quit the RSM Configuration application.
5. Launch the ANSYS Workbench.

```
runwb2
```

6. Open your workbench project file (.wbpj) copied to your scratch directory (**recommend to create the project file at your local machine where the same version of ANSYS is installed**).

**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 (**maybe the default one "Ansys Mechanical Enterprise"**).
- 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:

Properties of Schematic A5: Solution			
	A	B	C
1	Property	Value	Unit
2	General		
3	Component ID	Solution 5	
4	Directory Name	FFF-5	
5	Use Setup Launcher Settings	<input checked="" type="checkbox"/>	
6	Precision	Double Precision	
7	Display Mesh After Reading	<input checked="" type="checkbox"/>	
8	Embed Graphics Windows	<input checked="" type="checkbox"/>	
9	Use Workbench Color Scheme	<input checked="" type="checkbox"/>	
10	Load ACT Start Page	<input type="checkbox"/>	
11	Environment Path		
12	Initialization Method	Program Controlled	
13	Solution Monitoring	<input checked="" type="checkbox"/>	
14	Generate Solution Monitor Plots for Report	<input type="checkbox"/>	
15	Data Interpolation	<input checked="" type="checkbox"/>	
16	Generate Post Processing Images	<input checked="" type="checkbox"/>	
17	Notes		
19	Used Licenses		
20	Last Update Used Licenses	ANSYS Academic Research Mechanical and CFD	
21	Solution Process		
22	Update Option	Submit to Remote Solve Manager	
23	RSM Queue	Fidis-Debug	
24	RSM Queue Details		
25	HPC Configuration	Fidis	
26	HPC Queue	debug	
27	HPC Type	SLURM	
28	Job Name	Workbench	
29	Download Progress Information	<input checked="" type="checkbox"/>	
30	Progress Download Interval	120	s
31	Execution Mode	Parallel	
32	Number of Processes	8	
33	Restriction for Remote Design Point Update		
34	Serial Execution Only	<input type="checkbox"/>	
35	Specify Number of Processes Restriction	<input type="checkbox"/>	

- 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.

#### Command Line Execution of ANSYS Workbench in Batch Mode (useful in case of disconnection to cluster):

1. Save your project before Solve or Update.
2. Run the command as follows:

```
runwb2 -B -F WorkbenchProjectFile -R WorkbenchScriptFile &
```

It will execute the specified script, and shut down at the completion of script execution. Here is a sample script named as test.wbjn (note: can be generated by record journal in workbench):

```
# encoding: utf-8
```

```
# 2020 R2
```

```
SetScriptVersion(Version="20.2.221")
```

```
system1 = GetSystem(Name="FFF")
```

```
solutionComponent1 = system1.GetComponent(Name="Solution")
```

```
solutionComponent1.Update(Force=True)
```

```
Save(Overwrite=True)
```

```
exit
```

Note: With Force=True, the simulation continues from previous simulation. For a design point project, replace lines 4-6 by UpdateAllDesignPoints()

---

## Related articles

- [How to launch NAMD](#)
- [Data Transfer Nodes](#)
- [Managing a lab account](#)
- [Account management](#)
- [IMOD on Izar](#)